MicroStation/J™

Printing and Plotting Guide

BENTLEY

Trademarks

Copyrights

Table of Contents

Configuring Plotter Driver Files

Tutorial: Configuring Plotter Driver Files

Plotting Basics

Tutorial: Batch Plotting

Plot Resymbolization and Pen Tables

Tutorial: Plot Resymbolization Using Pen Tables

Plotting Using Image Manager

Appendix A: Plotting Key-ins

Appendix B: Plotter Driver File Syntax

1 Introduction

MicroStation's plotting tools provide many features that simplify the process of creating hard-copy plots of your designs.

Plotting with MicroStation

At various times during the design process most likely you will require plots showing part of, or the entire project. These may be in the form of scaled check plots or as "not to scale" illustrations for discussion purposes. Additionally, at the completion of the project, finished drawings will be required for construction or manufacture.

MicroStation provides the necessary tools that let you produce these hard-copy plots. At any stage during a project, you can create the required plots, from MicroStation, via the Plot dialog box.

Simple plotting vs. Complex plotting

Plotting can be as simple as setting up a view or placing a fence around the area of interest and clicking the Plot icon (see "Plotting at the simplest level" on page 5-1). Typically, the result will be a hard-copy plot of what you see on the screen.

Where the plot is to be at a particular scale, further setting up is required. For example, to produce a simple scaled plot of all or part of a view, you can set the scale in the Plot Layout dialog box (see "Plot layout criteria" on page 5-14). Creating completed scaled drawings that include a title block typically requires extra planning and setting up prior to plotting (see "Title block considerations" on page 5-3).

MicroStation also lets you create and use pen tables to resymbolize your design. In other words, you can produce plots that look different from the on-screen view of the design. Thus,

from a single design file, by using different pen tables, you can produce plots having different element symbology to that of the original design.

Configuration considerations

MicroStation lets you create a plot either:

- directly to a default system plotter/printer, or

- via a plotfile saved to disk for later submission to a plotter.

In both instances MicroStation uses a plotter driver file to correctly format the design data for your printer/plotter.

Selecting the correct printer or plotter driver

Most popular printers/plotters are supported by MicroStation with relevant plotter driver files. These are stored in MicroStation's "…\Workspace\system\plotdrv" directory.

You can select a plotter driver file, for your plotter, from the Plot dialog box (see "Choosing a plotter driver file" on page 3-2).

Selecting the correct plot size

Many plotters have pre-defined page sizes. These are defined in the plotter driver file. As a rule, both English and Metric page sizes are provided, with English sizes set as the default. You can edit the plotter driver file to change the units from English to Metric. This procedure is explained in a tutorial (see "Changing plotting units from English to Metric" on page 7-6).

Other options let you rotate the page 90°. As well, you can specify your own custom Width and Height values (see "Setting the page size and orientation" on page 5-12).

Selecting the correct scaling

You can set the scale for your plot from the Plot Layout dialog box. Two options are available that let you:

- set an exact scale for scale drawings (see "Setting the exact scale of a plot" on page 5-15).

- set the scale relative to the page size, which is useful for "not to scale" illustrations (see "Setting plot scale relative to the page size" on page 5-16).

When you are plotting drawings within a title block, other scaling factors such as text sizes also should be considered. These factors are discussed in "Title block considerations" on page 5-3.

Design resymbolization using a pen table

Often, the way you work "on-screen" does not produce easy to read plots. For example, you may use yellow and cyan colored lines on-screen because they are easy to see against a black background. When these same lines are plotted in color on white paper, the result is not easy to read.

MicroStation lets you create pen tables that, in turn, let you change the symbology of design file elements for plotting purposes. The design file is not changed in any way. The resymbolization occurs for the plotted output only. Using a pen table you can change the symbology of elements, including:

- Line Weight

- Color

- Fill Color

- Line Style

- Class

As well, you can specify text replacement, both for specific variables, such as filename and date, or for generic text items (such as a title in a title block).

Pen tables can be used to resymbolize any design file for plotting. Thus, you can produce several different plotted outputs from the one design file, without changing an element in the design. Chapter 11, "Plot Resymbolization and Pen Tables" covers these topics.

Plotting and reference files

Reference files are a valuable design tool and are fully supported by MicroStation's plotting tools. For simple plotting, you set up the view with all the required design elements displayed. You can then plot the view contents or place a fence around the area of interest.

Pen tables and reference files

With pen tables, you can selectively resymbolize elements from a particular reference file or from a list of reference files (see "Files" on page 11-10).

Raster-based reference files

MicroStation lets you attach raster reference files to your design. For example, this may be in the form of aerial photography of a region to which you overlay your design data. For such raster reference files to plot correctly, you must have a printer that is capable of plotting raster data. In general, these are the electrostatic, laser or inkjet type printers/plotters.

Plotting Image Manager raster files

Image Manager lets you manipulate image files of various formats. In order to plot these files, as with raster reference files, your printer must be capable of plotting raster data. Plotting from Image Manager is covered in chapter 13, "Plotting Using Image Manager".

Using print/plot preview

Using the Plot Preview window, you can review how the selected area will be plotted on the chosen printer. As you make changes to the plotting parameters, updating the preview window shows the effect of these changes on your plot. This can be a useful tool when you are creating a pen table, for example. You can view the results of the pen table resymbolization prior to plotting. This feature is covered in the tutorial contained in chapter 12.

Using plot configuration files

Plot configuration files let you save the plotting specifications for a design file. Once saved, you can recall the plot configuration file at any time to create a plot of the same area of your design file. You can create a plot configuration file that contains all parameters required to create a plot from a design file. These parameters include the area to be plotted, the levels that are displayed, page size, margins and scale. Where a pen table is attached, this too is saved as part of the plot configuration file.

When you require a plot during a design session, you can use the plot configuration file to set up the plot. In that case you do not have to remember, for example, which levels should be on or off for the plot. These settings, along with others, are saved with the plot configuration file.

For information on creating and using plot configuration files, see "Plot configuration files" on page 5-20.

Batch printing/plotting

BatchPlot is a utility for plotting and re-plotting related sets of design files by means of Job Sets and Plot Specifications.

- **Job sets** are used to identify, save, recall and print related sets of design files. See "Job Sets" on page 9-7.

- **Plot specifications** describe what to plot and how it should be plotted. See "Plot specifications" on page 9-2.

Using the BatchPlot utility you can set up groups of design files in Job Sets in order to plot automatically. Plotting specifications also are saved in the Job Sets. These let you define the Printer, Plot Area, Layout, and Display parameters. When batch plotting, these specification are applied to each design file in the list. BatchPlot lets you use shapes, cells or Views (including Saved Views) to define the area that should be plotted in each design file.

For more information on the BatchPLot utility, see chapter 9, "Batch Printing/Plotting".

2

Setting up Printers/Plotters

This chapter introduces you to the steps required in setting up a printer/plotter for use with MicroStation.

Introduction

MicroStation provides a variety of options for producing hard-copy plots of your entire designs or parts of designs from a printer/plotter. For example, you can

• Plot directly to a plotter connected physically to your PC.

• Plot directly to other "shared" plotters on a network.

• Create a plotfile that may be submitted to a plotter at another time.

Different plotters require different syntax for producing a plot. MicroStation provides a comprehensive range of plotter driver files that includes most commonly used printers/plotters.

With the Windows operating system, for example, you can attach a plotter to your own PC and use it to plot. Alternatively, you can access other plotters that are on the network. In order for you to use a printer, it must be installed. Where it is to be used by others on the network, it must be set to be "shared."

Installing a printer

You must install any printer before you can plot or print to it. In a networked environment, printers also can be shared. Sharing lets other users on the network print or plot to the shared plotter.

➤ **To install and (optionally) share a printer on Windows NT**

   1. From the Start menu, select Settings > Printers.

   The Printers window opens
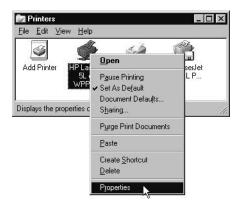
2. In the Printers window, click on the Add Printer icon.

The Add Printer Wizard appears.

3. Select My Computer and press Next.

4. Select the desired port.

5. Click the Configure Port button.

If you selected an LPT port, the Configure LPT Port dialog box opens.

If you selected a COM port, the Ports dialog box opens.

6. Configure the selected port.

7. Click Next on the Add Printer Wizard dialog box.

8. Select the appropriate manufacturer and printer, then click Next.

When you send a device-ready file to a Windows printer queue, you are not really using a Windows print driver. However, Windows still requires that you enter a manufacture and printer.

9. Accept the default printer name or enter one of your own in the Printer Name field. (Use a maximum of 12 characters if you want to share your printer with Windows 95 systems.)

You can make this printer the default for Windows-based programs by clicking the Yes button.

10. Click Next.

11. (Optional) To enable this printer to be shared with other networked users, click Share. Then select the operating system of the computers sharing the printer.

12. Click Next.

13. (Optional) Turn on Yes to print a test page or No to bypass.

14. Click Finish to complete the setup procedure.

✍ On Window NT, you must make sure the default datatype in the Print Processor dialog box is set to RAW. RAW is the default but some products require you to change this setting. You can access the Print Processor dialog box by right clicking on the printer icon, then select Properties. Click the Print Processor button to open the Print Processor dialog box.

✍ If you give your shared printer a name that is greater than 12 characters, this printer will not be "seen" on Windows 95 systems.

You can share a printer that is not already shared. That is, a printer that has been installed on your system but not currently shared.

➤ **To share an existing installed printer in Windows NT**

1. From the Start menu, select Settings > Printers.

The Printers window opens. In this window are icons for all installed printers.

2. Right click on the icon for the printer you want to share.

3. From the menu that opens, select Properties.

The properties dialog box for the selected printer opens.

4. In the Properties dialog box, click the Sharing tab.

5. Turn on Shared then select the operating system of the computers sharing the printer.

6. Accept the default printer name or enter one of your own in the Printer Name field. (Use a maximum of 12 characters if you want to share your printer with Windows 95 systems.)

7. Click the OK button.

The Properties dialog box closes and the printer is shared.

Using similar procedures to those for Windows NT, you can install and share printers in the Windows 95 system.

➤ **To install a printer on Windows 95**

1. From the Start menu, select Settings > Printers.

2. Click on the Add Printer icon.

The Add Printer Wizard appears.

3. Click Next.

4. Select Local Printer and press Next.

5. Select the appropriate manufacturer and printer, then click Next.

   When you send a device-ready file to a Windows printer queue, you are not really using a Windows print driver. However, Windows still requires that you enter a manufacture and printer.

6. Select the desired port.

7. Click the Configure Port button.

   If you selected an LPT port, the Configure LPT Port dialog box opens.

   If you selected a COM port, the COM1 Properties dialog box opens.

8. Configure the selected port and click OK (in the Configure Port dialog box).

9. Click Next on the Add Printer Wizard dialog box.

10. Accept the default printer name or enter one of your own in the Printer Name field.

    You can make this printer the default for Windows-based programs by clicking the Yes button.

11. Click Next.

12. (Optional) Turn on Yes to print a test page or No to bypass.

13. Click Finish to complete the setup procedure.

In order to share your Windows 95 printer with other users, you have to turn on Sharing for the printer.

➤ **To share your Windows 95 printer**

1. Click the icon for the printer that you want to share.

2. From the File menu, choose Properties.

3. Click the Sharing tab.

4. Turn on Shared As.

5. Click OK.

Accessing a networked printer

When a networked printer is installed on another PC on the network and set to be shared, you can access that printer and plot to it from your PC. You can submit your plots to the networked printer:

- Directly, via a customized plotter driver (see "Using the plotter driver file" on page 2-5).

- Using the "printer.plt" plotter driver file. With this method, the networked printer must be installed on your system (see "Using "printer.plt" plotter driver file" on page 2-5).

- By creating a plotfile on disk and then copying this to the printer in a DOS window.

Using the plotter driver file

You can access a networked printer, via the "`default_outFile`" record of the plotter driver file. For example, if you specify the following:

```
default_outFile=\\PrinterPC\printer
```

then plots will be sent to the printer "`printer`" attached to the PC "`PrinterPC`".
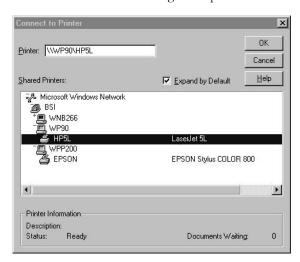
Using "printer.plt" plotter driver file

If you want to access a networked printer using the "printer.plt" plotter driver file, then you must first install the printer on your system. This printer then can be set as the default system printer.

➤ **To add a networked printer on Windows NT**

1. From the Start menu, select Settings > Printers.

2. Click on the Add Printer icon.

   The Add Printer Wizard appears.

3. Select Network printer server and press Next.

The Connect to Printer dialog box opens.
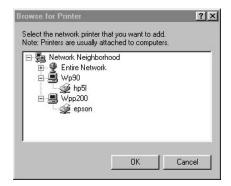
4. Select the required printer (usually printers are attached to computers so click on the computer to display the attached printer).

5. Click OK.

6. Select Yes or No to define whether you want your Windows-based programs to use this printer as the default, then click Next.

7. Click Finish to complete the setup procedure.

➤ **To add a networked printer on Windows 95**

1. From the Start menu, select Settings > Printers.

2. Click on the Add Printer icon.

The Add Printer Wizard appears.

3. Click Next.

4. Select Network Printer and click Next.

5. In the Network path or queue name field, key in the path name for the required printer and go to step 8. If you are not sure of the path, click Browse and continue with the next step.

The Browse for Printer dialog box opens.

6. Select the required printer (printers are usually attached to a computer, so click the computer to display the attached printer).

7. Click OK.

The Browse for Printer dialog box closes and the network path for the printer appears in the Network path or queue name field.
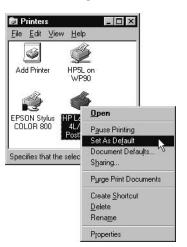
8. Click Next.

You are given the option of choosing a new name for the printer or accepting the default.

9. Select Yes or No to define whether you want your Windows-based programs to use this printer as the default, then click Next.

10. Select Yes or No to print a test page, then click Finish.

➤ **To set a networked printer as your default system printer**

1. From Windows Start menu, select Settings > Printers.

2. Right-click on the icon for the required printer and from the menu, select Set As Default.

This toggles the setting. If you re-open the menu, you will see a checkmark against Set As Default.

Setting a networked printer as the default system printer.

With the networked printer set as the default system printer, you can plot directly to it from MicroStation using "printer.plt" as the plotter driver file.

✎ You can plot directly to other networked printers using "printer.plt" even if they are not the default system printer. When you select Page Setup, from the Plot dialog box, you can select another networked printer along with the page size and orientation.

Creating a plotfile and copying to the plotter

With this method, plots are not submitted directly to the plotter. Plotfiles for the selected plotter are saved to disk for subsequent submission to the plotter via a DOS session window.

When the selected plotter is neither "printer.plt" nor the system printer, then you can create plotfiles and save them to disk. These plotfiles contain all the data required to create the plot on the selected plotter. They can be copied to the plotter from a DOS session window using the syntax:

```
copy /b <filename> <printer>
```

Plotfiles created from the same design file have the same name with an incremental suffix, such as "mydesign.001," "mydesign.002" and so on. In these instances, you can use a wildcard to submit all plots with the one key-in, such as:

```
copy /b mydesign.* \\printman\A0plotter
```

3

Configuring Plotter Driver Files

This chapter covers the selection and modification of the plotter driver files that are delivered with MicroStation.

Introduction

When preparing to plot your design, an important consideration is the selection and configuration of the appropriate plotter driver file. MicroStation includes a directory containing sample plotter driver files, "…Workspace\system\plotdrv." This directory contains plotter driver files for most popular printers and plotters. In most instances, you will be able to select a prepared driver file that is correctly configured for your plotter. It is possible also, that you may discover there is no driver file for your plotter or that the supplied driver file requires modification. In these cases, the information in this chapter will guide you through the process of reconfiguring the supplied plotter driver to suit your requirements. Alternatively, it will help you in creating a plotter driver file suitable for your plotter.

Choosing a plotter driver file

Depending on the plotter that you will use, you have several choices when choosing the plotter driver best suited for your needs.

1. If your plotter or printer uses a windows-based driver designed especially for your device, you can use the "printer.plt" driver. This plotter driver file is intended to interface with the Windows 95/98 or Windows NT printer driver setup. Usually, if you are printing in a network environment, this driver should be used.

2. If your plotter or printer is a Post Script-based device or supports Post Script printing, there are several drivers especially for that purpose.

3. If your printer or plotter supports HPGL/RTL printing or plotting and you will be plotting design files containing both vector and raster information, you should use one of the available plotter drivers intended for printers that support HPGL/RTL.

Other considerations are whether or not your printer/plotter is connected to a network or connected directly to your system via a parallel or serial port. You should take into account, also, whether or not you will first create a plot file before sending the file to the printer.

You may not need to customize or modify the driver that you will use. MicroStation includes about 60 sample printer/plotter drivers in the …\Workspace\system\plotdrv directory. It is likely that you will find a plotter driver that either is intended for or works with your plotter.

Once you determine the category of printer or plotter driver that you need, you can select an appropriate driver. For some plotters that have both monochrome and color models, there are both color and monochrome versions of the driver.

➤ **To choose a plotter driver file**

1. In the Plot Preview dialog box, click the Plotter Driver icon.

   or

   From the Setup menu, choose Driver.

   The Select Plotter Driver File dialog box opens.

2. From the list of supplied drivers, in MicroStation's "…\Workspace\system\plotdrv" directory, select the required plotter driver.

3. Click OK.

The Select Plotter Driver File dialog box closes.

Supported plotters/printers

MicroStation's plotting system generates output in formats supported by most plotting/printing devices. There are three types of plotter driver parameter (.plt) files that are delivered with MicroStation:

- generic templates such as "cal907.plt," "hpgl2.plt," "pscript.plt" and "epson24.plt."

- customized parameter files such as "cal524xx.plt," "xerox8830.plt" and "hp650c.plt."

- "printer.plt," which works in conjunction with the System Printer driver provided by the output device's vendor.

All delivered MicroStation plotter driver files reference drivers that create plot information in industry-recognized formats (such as CAL907/PCI, HPGL/2, HPGL/RTL, ESC/P, TIF, CGM and SVF). If a specific ".plt" file does not exist for the device that you are using, you may be able to use another existing ".plt" file. Alternatively, you can create one yourself.

Following is a table showing the plotter languages for which sample plotter driver files are supplied and the corresponding filenames.

Plotter language (driver)	Plotter Driver File(s)
Calcomp 906	cal906.plt
Calcomp 907	cal104x.plt, cal524xx.plt, cal906.plt, cal907.plt, ver8536.plt, ver8524.plt
Calcomp 960	cal960.plt
DM/PL (Houston Instrument)	hidmp40.plt, hidmp52.plt, hidmp56.plt, ioline.plt
Enhanced Windows Metafile	emf.plt
ESC/P	epson8.plt, epson8h.plt, epson24.plt
HP-GL	drftpro.plt, mutoh500.plt, drftmstr.plt, hp7470a.plt, hp7440a.plt, hp7475a.plt, hp7550a.plt, hp7580b.plt, hp7585b.plt
HP-GL/2	hpgl2.plt, hpljet3.plt, hpljet4.plt, hpljet4v.plt, hpdjet.plt, hpxl300.plt, hp650c.plt, drftprop.plt, novajet2.plt,
HPGL/RTL	xerox8830.plt, oce_rtl2.plt, hpglrtl.plt

Plotter language (driver)	Plotter Driver File(s)
PCL	hpljet.plt, hp1200c.plt, hp5xxc.plt, hppcl5.plt
PostScript	pscript.plt, pscriptc.plt, ps650c.plt, psc650c.plt
SVF	svfhires.plt, svflores.plt
Raster File	jpeg.plt, png.plt, tiff.plt, cals.plt

When you submit a plot, its production is based on the specifications in the loaded plotter driver file. By default, MicroStation uses the most recently used ".plt" plotter driver file.

Plotter driver files

For each supported plotter, MicroStation supplies a sample **plotter driver file** — a text file that defines all the necessary information to create a plotfile. The plotter driver file specifies the following:

- Plotter model

- Number of pens the plotter can use

- Resolution and units of distance on the plotter

- Pen change criteria

- Name, size, offset and number for all paper sizes

- Stroking tolerance for arcs and circles

- Border around plot and information about border comment

- Pen speeds, accelerations and force where applicable

- Pen to element color or weight mappings

- Spacing between multiple strokes on a weighted line

- Number of strokes generated for each line weight

- The definitions for custom line styles (on plotting only)

- The method by which plots are generated

- Actions to be taken at plot's start, end and on pen changes

You can use any text editor to create or to customize plotter driver files. All plotter driver file records and the required syntax are listed in "Plotter Driver File Syntax". The quickest way to customize a plotter driver file is to edit one of the sample files

supplied with MicroStation. For typical examples, see
"Customizing plotter driver files" on page 3-11).

✍ If you modify a sample plotter driver file, it is a good idea to
retain the original file and to save the modified file as a new file
with a different name, in a separate directory. MicroStation
provides you with a separate directory for this purpose —
"…Workspace\standards\plotdrv." As well as customized plotter
driver files, you could use this directory to store copies of all
plotter driver files that you commonly use.

When the plotter driver file is used

The plotter driver file is used by MicroStation when a plotfile is
created and in some cases, by the subsystem that "transports" the
plotfile to the plotter.

✍ For a plot to reflect all changes made to a plotter driver file, you
usually must generate a new plotfile in MicroStation. (The
exception is a change to the communications settings such as
baud rate, parity and stop bits that are used by the "transport"
subsystem to get the plotfile to the plotter.)

Plotter driver file format

Plotter driver files contain records and associated keywords that
can be entered in any order (with a few noted exceptions). The
maximum record length is 80 characters, so each line is a record.
Where a record requires more than 80 characters, you can place a
backslash (\) at the end of the line and continue on to the next
line. The backslash acts as a continuation character. Multiple lines
can be included in a single record by placing backslashes at the
end of each preceding line. For example:

    weight_strokes(mm)= (0.25, 0.50, 0.75, 1.00, 1.25, 1.50,\
    1.75, 2.00, 2.25, 2.50, 2.75, 3.00, 3.25, 3.50, 3.75,\
    4.00, 4.25, 4.50, 4.75, 5.00, 5.25, 5.50, 5.75, 6.00)

is read as a single record although it spans several lines.

Each record specifies one or more of the parameters described in
the following sections.

Conventions

These conventions must be followed:

- Records can be in either uppercase or lowercase letters.

- Spaces and tabs before and after data are ignored. All characters to the right of a semicolon are treated as comments and are ignored.

- Blank records and comment-only records are permitted.

- Keywords can be abbreviated to the minimum number of characters required to uniquely identify them. However, it is recommended that you avoid abbreviations so that the addition of keywords to future releases of MicroStation will not make previously unique keywords ambiguous.

Included files

Using the `%include` statement in your plotter driver file lets you include another file containing plotter driver information. Typically, the include file will not be a complete ".plt" file but instead will contain one or more records that can be shared by other "parent" files. Multiple %include statements may be used to include multiple files. Included files, in turn, may include other files.

Exceptions

A file cannot include itself nor may it include another file more than once. A file cannot include a file that would cause the inclusion process to loop back on itself. For example, if File A includes File B and File B attempts to include File A, then File B's `%include` statement is ignored.

Example

Take, for example, the `style` keyword. This now supports device-independent units. Therefore, definitions could be moved out of your plotter driver files and into a shared file, which could be included in all your plotter driver files. You then can modify the style definitions for all your plotters by editing this one shared file.

Configuring your plotter driver

MicroStation's plotter driver directory includes drivers for the most common printers and plotters. Each of these is configured to contain only the settings and parameters supported by the associated printer or plotter. For example, plotters such as the HP 7475A do not support raster plotting nor do they accommodate larger paper sizes. In addition, these plotters do not support features such as polygon element filling. Therefore, they have limited parameter values for these and other specific parameters.

Instructions for configuring each plotter are contained in the relevant plotter configuration file. Pictured below is part of the "hpglrtl.plt" plotter configuration file, which has been opened in Windows Notepad. It shows the Size and Style records. Along with the settings are instructions for their use.

```
hpglrtl.plt - Notepad                                         _ □ ×
File  Edit  Search  Help
; The SIZE records defines the plotting area of page.  The .27 offset
; given is to allow room for the border record.
; Use the first set when using roll paper and the second set for cut paper.
; Only 5 SIZE records can be defined at one time

; ENGLISH SIZE records
size=(8.5,11.0)/num=0/off=(0.27,0.27)/name=a
size=(11.0,17.0)/num=0/off=(0.27,0.27)/name=b
size=(17.0,22.0)/num=0/off=(0.27,0.27)/name=c
size=(22.0,34.0)/num=0/off=(0.27,0.27)/name=d
size=(34.0,44.0)/num=0/off=(0.27,0.27)/name=e
resolution(IN)=(0.00098425197,0.00098425197)    ; specifies both res and units

; ENGLISH SIZE records
; size=(6.28,9.5)/num=0/off=(0.27,0.27)/name=a         ; cut sheet
; size=(9.0,14.0)/num=0/off=(0.27,0.27)/name=b         ; cut sheet
; size=(14.0,20.0)/num=0/off=(0.27,0.27)/name=c ; cut sheet
; size=(20.0,31.0)/num=0/off=(0.27,0.27)/name=d ; cut sheet
; size=(42.00,32.0)/num=0/off=(0.27,0.27)/name=e       ; cut sheet

; Metric size records
; size=(158,244)/num=0/off=(0,7)/name=a4
; size=(229.0,355.0)/num=0/off=(0,7)/name=a3
; size=(355.0,508)/num=0/off=(0,7)/name=a2
; size=(788,508)/num=0/off=(7,7)/name=a1
; size=(1066,813)/num=0/off=(7,7)/name=a0
; resolution(MM)=(0.025,0.025)                  ; Metric resolution

;  The style records defines how the design file line codes (styles)
;  are to be plotted.  Values are in plotter units (resolutions).
;  Values determine pen down/up movements, where (14,42) for a dot
;  pattern leaves the pen down for 14 units and up for 42 units.  The
;  /nohardware switch causes software stroking with these values.

style(1) = (14,42)/nohardware        ; style = dot
style(2) = (70,42)/nohardware        ; style = med dash
style(3) = (168,56)/nohardware       ; style = long dash
style(4) = (112,42,28,42)/nohardware   ; style = dot-dash
style(5) = (56,56)/nohardware        ; style = short dash
style(6) = (84,28,28,28,28,28)/nohardware      ; style = dash-dot-dot
style(7) = (112,28,56,28)/nohardware           ; style = long dash - short dash
```

Part of the "hpglrtl.plt" plotter configuration file.

For specific information and guidance in configuring and using the driver associated with your plotter, open the plotter file that you intend to use in an appropriate text editor, such as Windows Notepad. You can then follow the instructions in the file to configure your plotter driver file. Several common customization tasks are discussed in "Customizing plotter driver files" on page 3-11.

Configuring PostScript Drivers

You can use MicroStation's PostScript plotter driver files to create PostScript and Encapsulated PostScript files. Postscript printer and plotter driver files are configured similarly to other printer and plotter driver files. However, there are several minor differences to which you should give special attention to ensure that your driver is correctly configured.

The following PostScript plotter driver files are supplied:

- "pscript.plt" (for monochrome)
- "pscriptc.plt" (for color)
- "ps650c.plt" (for large format monochrome)
- "psc650c.plt" (for large format color)
- "epscripm.plt" (monochrome Encapsulated PostScript)
- "epscripc.plt" (color Encapsulated PostScript)

Both the Encapsulated PostScript plotter driver files optionally add a preview/thumbnail image to the PostScript files that they produce. Some applications use these thumbnails to preview the PostScript image.

By default, the prolog file "pscript.pro" is used to provide the header for the PostScript file. You can edit the prolog file, which also is a text file, to change default settings such as which paper tray is used. (You can also edit the `prolog` record in the plotter driver file to specify a different prolog file.) The settings in the prolog file are dependent on the plotting driver; refer to your plotter's programming manual for the correct syntax.

Each of the supplied prolog files in the "hpt*.pro" series — for example, "hptlegal.pro" — specifies a particular paper tray for the HP LaserJet 4v laser printer.

HPGL-2/RTL plotters

Most plotters that support HPGL-2/RTL work with the "hpglrtl.plt" plotter driver file. The HPGLRTL driver produces HPGL-2 vectors from vector data sources and RTL rasters from raster data sources.

Many devices that support HPGL-2/RTL have numerous limitations. For example, these devices do not support both HPGL-2 vectors and RTL rasters in the same plot. Nor do they support all the different RTL pixel formats and color depth. In addition, they may not support all of the RTL compression modes described in the RTL specification from Hewlett Packard.

Compression Modes

RTL compression modes are defined by the `/methods` option. Modes supported by the HPGLRTL driver are:

Compression Method 0 — None (unencoded).

Compression Method 1 — Run Length Encoding.

Compression Method 2 — TIFF PackBits Encoding.

Compression Method 3 — Delta Row (Seed Row) Encoding.

Compression Method 4 — Empty Row (valid only if `/format=2`).

Compression Method 5 — Duplicate Row (valid only if `/format=2`).

You can use the `raster_compression` record in "hpglrtl.plt" to restrict the compression modes (that the driver uses) in devices that do not support all RTL compression modes. The default settings for this record prevent compression to ensure that the driver output functions with any device that claims HPGL-2/RTL support.

If you want to minimize driver output, you can experiment with different plotter settings. The `raster_compression` keyword controls the overall format of compressed data. The two formats are:

1. (Default) `raster_compression/format=1/methods=(0)`

2. `raster_compression/format=2/methods=(2,3,1,0,4,5)`

Format 1

Use this line for no compression. Format 1 indicates that raster data must be transferred by row. This format is more likely to work with any plotters that claim RTL support.

Format 2

This commented out line configures full adaptive compression. It is the most aggressive setting and results in the smallest driver output. Use this line for full adaptive compression. It indicates that raster data may be transferred in RTL blocks.

✍ The integers assigned to the "methods" keyword determine which supported compression methods are used by the driver during adaptive compression. You can specify a sole compression method if a device does not support adaptive compression.

The Simple Vector Format Driver

Simple Vector Format (SVF) is a vector graphics format that supports hyperlinks and layer information for use on the World Wide Web.

This driver creates a viewable SVF file. It is not intended to be used to scale a physical drawing. Hence, any reference to margins or drawing setup are not applicable.

This format is available in either of two files:

- svfhires.plt
- svflores.plt

svfhires.plt

This driver file has a fixed record size. The SVF output from this file is scaled to use the entire SVF coordinate system (16-bit). The high resolution allows you to zoom in using an appropriate SVF Netscape plug-in. It produces files of a larger size than the svflores.plt file and takes longer to generate, transmit and display the files.

svflores.plt

This driver has a variable record size corresponding to typical display resolutions. The "svflores.plt" files are smaller and quicker to display and create than those from "svfhires.plt", but have limited zoom capabilities at the lower resolution. If you are intending to publish a picture on the web, you might want to use low or high resolution SVF files depending on their circumstances.

✍ SVF image quality is resolution-dependent for MicroStation design files, since there is no support for line styles.

✎ Line weights vary depending on your zoom level.

✎ Dashed lines are exploded and also vary with zoom level.

Customizing plotter driver files

Not everyone wants the same default settings for plotting as those in the delivered plotter driver files. When you want something that is different from these delivered default settings, you can modify your plotter driver file to suit your requirements.

Plotter driver files are ASCII text files and should be modified with an ASCII text editor, such as Notepad that comes with Windows 95 and Windows NT. This is a simple text editor that is opened by selecting:

Start > Programs > Accessories > Notepad

In the following pages, commonly used customizing tasks are discussed.

✎ Whenever you modify a delivered plotter driver file, it is recommended that you save the modified version under a new name, and place it in the "…Workspace/standards/plotdrv" directory. This reduces the chance of your customized plotter driver files being deleted or overwritten during a product re-install or update.

Defining default plot settings

When you first select a plotter driver from the Plot dialog box, a number of the plotter driver file records control settings in this dialog box. That is, the records define default settings that can be changed, temporarily, from within the Plot dialog box. They apply only for that session. The next time that you select that plotter driver file, the settings revert to the values defined in the

plotter driver file. Records that fall into this category include those relating to the following:

- "Default sheet size" on page 3-13
- "English/Metric units" on page 3-12
- "Default scale" on page 3-14
- "Rotate the plot on the page" on page 3-14
- "Center plot on page" on page 3-14
- "Default plot border" on page 3-15
- "Default pen table" on page 3-15
- "Forcing all colors to black on a color/grayscale printer" on page 3-15
- "Defining plotted line thicknesses" on page 3-16
- "Accessing specific Windows system printers" on page 3-16
- "Mapping PostScript fonts to MicroStation fonts" on page 3-17

You could create a plot configuration file containing the settings that you require, but you would have to remember to load this file prior to plotting. Another method is to modify the plotter driver file for your plotter to reflect the your preferred default settings. This modified plotter driver file could be saved under a name that reflects its purpose. For example, you may do a lot of plotting on A0 sheets at 1:100 scale. You could modify the plotter driver file accordingly and save it under a name such as "A0_100scale.plt." In future, when you want to create a plot on A0 sheet at 1:100 scale, you could select this plotter driver file.

English/Metric units

By default MicroStation's delivered plotter driver files are set up with English units as the default. Metric units are provided as well, but they are commented out (with a semi-colon preceding each record). To change from english to Metric, you simply place a semi-colon (;) prior to each record for the English units and delete the semi-colon preceding each record for the Metric units.

When you are using the Windows system printer (printer.plt) the units are determined by the Windows' Measurement system setting. To change the units, use the Regional settings dialog box, not the plotter driver file. Units are set in the Numbers tab of the Regional Settings dialog box.

➤ **To change plotter units for Windows system printer (printer.plt)**

1. Select Start > Settings > Control Panel to open the Control Panel window.

2. Double-click the Regional Settings icon.

3. Click the Number tab.

4. From the Measurement system option menu choose from Metric or US (that is, English units).

Default sheet size

As a rule, plotters are capable of plotting on sheets of various sizes. You can select the required sheet size from the Page Size option menu in the Page Setup dialog box.

MicroStation's delivered plotter driver files contain records for both English and Metric sheet sizes, with English set as the default. Available sheet sizes are listed in the `size` records. By default, the last sheet in the list is selected as the default sheet size. To make another sheet size the default, simply edit the plotter configuration file to place the required size last in the list. For example, the plotter driver file "psc650c.plt" has size records for English sheets A, B, C, D, and E as shown below:

```
; ENGLISH resolution and SIZE records
size=( 6.9, 9.4)/num=0/off=(.8,.8)/name=A
size=( 9.4,15.4)/num=0/off=(.8,.8)/name=B
size=(15.4,20.4)/num=0/off=(.8,.8)/name=C
size=(20.4,32.4)/num=0/off=(.8,.8)/name=D
size=(32.4,42.4)/num=0/off=(.8,.8)/name=E
resolution(IN)=(0.003333333333333333,0.00333333333333333
3) ;300DPI
```

In the Page Setup dialog box, E, the last page size record listed, is the default selection on the Page Size option menu. If you wanted C to be the default, you simply edit the plotter configuration file to place the record for C after that of E, thus:

```
; ENGLISH resolution and SIZE records
size=( 6.9, 9.4)/num=0/off=(.8,.8)/name=A
size=( 9.4,15.4)/num=0/off=(.8,.8)/name=B
size=(20.4,32.4)/num=0/off=(.8,.8)/name=D
size=(32.4,42.4)/num=0/off=(.8,.8)/name=E
size=(15.4,20.4)/num=0/off=(.8,.8)/name=C
resolution(IN)=(0.003333333333333333,0.00333333333333333
3) ;300DPI
```

This would result in C being the default setting on the Page Size option menu in the Page Setup dialog box.

Default scale

Normally, plots are produced at a particular scale. This can be set prior to plotting in the Plot Layout dialog box. In this dialog box, you can specify the scale of your plot as a ratio of MU:SU to Plotter Units. Consider, for example, your design file having Working Units of Feet (MU) and Inches (SU), while your plotter's units are Inches.

To create a 100 feet to the inch scale plot, you would enter 100 in the *Scale to ft:in/IN* field. That is, the plot would be 100 ft per IN.

You can use the `/scale` option for the `size` record to specify a scale factor for any sheet size. Take, for example, the situation where your design files were set up with feet as the master units, and you always created 100 feet to the inch plots on E size sheets. In your plotter driver file, you could modify the E size record to be:

```
size=(32.4,42.4)/num=0/off=(.8,.8)/scale=100/name=E
```

Whenever you selected E size sheets for the modified plotter driver, the default scale would be 100 feet to an inch.

Rotate the plot on the page

Used to enable/disable automatic rotation to maximize plots. This setting in the Page Setup dialog box is controlled by the `rotate` record of the plot configuration file. By default, it is usually set to `none` (no rotation). Other options, `cw` and `ccw` let you rotate the plot 90° clockwise or counter-clockwise, respectively. For example, to rotate plots 90° clockwise, you should change `none` to `cw`, thus:

```
rotate=cw
```

Center plot on page

This setting in the Plot Layout dialog box is controlled by the `autocenter` record of the plot configuration file. To turn this setting off so that the plot is not automatically centered on the page, you simply place a semi-colon (;) at the beginning of the record:

```
;autocenter
```

A semi-colon preceding a record "comments out" the line and causes it to be ignored.

Default plot border

By default, the Plot Border option is enabled in plot driver files. This setting can be turned off from the Plot Options dialog box. Alternatively, you can disable the plot border by commenting out the `border` record. To do this, place a semi-colon at the start of the `border` record (that is, the line in the plotter driver file beginning with the word "border"). When you select the plotter driver file with this record disabled, the Plot Border setting in the Plot Options dialog box is turned off by default.

Default pen table

Where you have a pen table that is used for all plots in a project, you can have it loaded, automatically, when you select the plotter driver file. By default, the `pentable` record is commented out and appears in the plot driver files as follows:

```
;pentable=\dir\file.tbl
```

Take the case, where you use a pen table "civproj.tbl," which is in the directory "d:\pentables." You can configure your plot driver file to have this pen table loaded automatically. You do this by removing the semi-colon and adding the pen table directory and name as follows:

```
pentable=d:\pentables\civproj.tbl
```

When you select the plotter driver file with the above record enabled, the pen table also is loaded.

Forcing all colors to black on a color/grayscale printer

When you create designs in MicroStation, it is normal to use colored lines to help discern the various elements of the design. When these designs are plotted on a color printer/plotter, the lines are plotted in the same colors, which may make them hard to read. This applies particularly to colors such as yellow and cyan, which are easily seen against the black background of your screen but not so easily seen against the white paper background.

Typically, plotter driver files for color printers have the following setting:

```
num_pens=255
```

to accommodate the colors in MicroStation's color palette. Changing this setting to:

```
num_pens=1
```

forces all colors to the first color — black.

Defining plotted line thicknesses

MicroStation lets you map design file line weights to plotted line thicknesses using physical units of millimeters (mm) or inches (in) or by plotter resolution units (dots - the default). The plotter driver file record that maps MicroStation line weights to plotted line thicknesses is `weight_strokes`. In the past, this setting was dependent on the plotter resolution units (dots) but now you can use physical units. This makes it much easier to read the plotter driver file and the same settings can be used by any plotter.

For an example, let us say that you use only eight line weights (0 to 7) in your designs. In the plotted output, you want these line weights to be mapped to the following eight line thicknesses (measured in millimeters):

0.15, 0.25, 0.35, 0.50, 0.70, 1.00, 1.50, 2.00

To do this, you simply change the `weight_strokes` setting to the following:

```
weight_strokes(mm)=(0.15, 0.25, 0.35, 0.50, 0.70, 1.00, 1.50,
2.00)
```

If you use more line weights/thicknesses than will fit on one (record) line, then place a backslash (\) at the end of the line and continue on the next line. This lets you create a record that spans two or more lines. For example, to map all of MicroStation's 32 line weights to plotted line thicknesses, you may have the following:

```
weight_strokes(mm)= (0.25, 0.50, 0.75, 1.00, 1.25, 1.50\
1.75, 2.00, 2.25, 2.50, 2.75, 3.00, 3.25, 3.50, 3.75\
4.00, 4.25, 4.50, 4.75, 5.00, 5.25, 5.50, 5.75, 6.00)
```

Accessing specific Windows system printers

When you select the delivered file "printer.plt" as your plotter driver file, plots are sent to the default Windows system printer. The default printer is specified by the `sysprinter` record. If no name is supplied, then the default Windows system printer is used. However, you can configure this plotter driver file to send plots to other accessible printers on your network. The recommended way to do this is to create a copy of "printer.plt" with the other printer named in the `sysprinter` record.

For example, to access a system plotter named "LF Plotter" shared from the PC "PlotMan2," you could copy "printer.plt" to be "lformat.plt" and edit its `sysprinter` record to be:

```
sysprinter = "\\PlotMan2\LF Plotter"
```

Additionally, you can specify a default "form" (sheet size), orientation (portrait or landscape) and offset (to leave space at the bottom of the plot so that border text may be included). For example:

```
sysprinter = "\\PlotMan2\LF Plotter"/form=A0/
orientation=landscape
```

instructs the plotting process to use an A0 size sheet with landscape orientation.

Mapping PostScript fonts to MicroStation fonts

Where you are plotting to a PostScript device that has resident fonts, you can map these fonts to MicroStation fonts in the plotted output. However, the PostScript font must be resident in the printer prior to sending the plotfile to the printer. In the plotter driver file, you can add a record for `font` to map the required PostScript font to one or more fonts in the design file. Differences in the PostScript font compared to the design file font may require adjustments with the `/Kern` and `/Sizeratio` parameters to make the new font appear at a similar size and spacing.

For example, the following:

```
font("Helvetica")=(1-4)
font("Courier")=(3)
```

maps MicroStation fonts 1, 2 and 4, to the Helvetica font and maps MicroStation font 3 to the Courier font on the PostScript printer.

In the following:

```
font("Helvetica")=(1-4)/Kern=35/Sizeratio=1.5
```

maps MicroStation fonts 1-4 to the Helvetica font but the characters will appear at 1.5 times their normal size with intercharacter spacing (kerning) set at 35.

Refer to "font" on page B-8 for more information.

4   Tutorial: Configuring Plotter Driver Files

This tutorial introduces you to the principles of configuring plotter driver files.

Introduction

Plotter driver files are ASCII text files that contain instructions for the selected plotter. You can create plotfiles to disk for later plotting or you can send the plot directly to an attached plotter. In either case, settings in the plotter driver file control the plotted output.

Using a plotter driver file

By default, the plotter driver files are set up with the largest page size for a plotter as the default. Also, the default orientation is portrait. You can change these settings in the Plot Options dialog box. Alternatively, you can set up your plotter driver file with the default settings as you prefer. Where you have a "non-standard" page size, you can configure this also.

In the following exercises, we will be working with the "ps650c.plt" plotter driver file. This file is configured for a large format monochrome PostScript plotter.

➤ **Prepare for the exercise**

1. Open the design file "office.dgn," which is located in MicroStation's "Workspace\projects\tutorials\intro\dgn" directory.

   This file has been saved with View 1 (only) open.

2. Use the *Fit View* view control to fit the view.

3. From the File menu, choose Print/Plot.

   The Plot dialog box opens.

4. In the Plot dialog box, click the Plotter Driver icon.

The Select Plotter Driver File dialog box opens.

5. From the list of Files, choose ps650c.plt.

6. Click OK.

7. In the Plot dialog box, click the Preview Refresh icon.

The Plot dialog box expands to display the plot preview window and the dialog box's name changes to Plot Preview.

The Plot Preview dialog box showing the proposed plot.
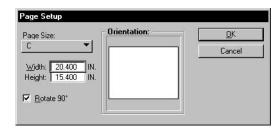
Let us assume that you are required to create a plot on an English C sized sheet. Also, the aspect ratio of the drawing is "landscape" while that of the displayed sheet is "portrait." These can be changed as follows.

➤ **Change the page size and orientation**
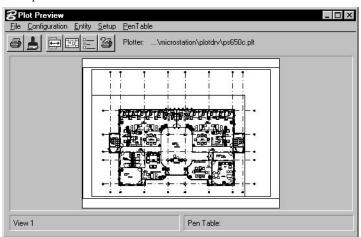
1. In the Plot Preview dialog box, click the Page Setup icon.

or

From the Setup menu, choose Page.

The Page Setup dialog box opens showing that the Page Size is "E."

2. From the Page Size option menu, choose C.

3. Turn on Rotate 90°.

4. Click OK.

The Page Setup dialog box closes. The preview window updates to show the new sheet size and orientation.

Plot Preview dialog box after changing the sheet size and rotation.

As it stands, you would have to make these changes every time that you wanted to create a C sized plot with landscape orientation. A simpler method is to create a plotter driver file that has your required settings as the defaults.

Before continuing, return the plotter driver settings back to the defaults. This is done by reselecting the plotter driver.
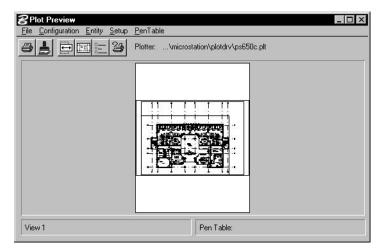
➤ **Return to the default settings**

1. In the Plot dialog box, click the Plotter Driver icon.

The Select Plotter Driver File dialog box opens.

2. From the list of Files, choose ps650c.plt.

3. Click OK.

The preview window updates to display the proposed plot with the default plotter driver file settings.

Changing the plotter driver's default settings

MicroStation's plotter driver files are ASCII text files. They contain all the required settings for the selected plotter. For this exercise, we will look at the sheet size and rotation records. In the exercise we will be referring to the Windows text editor, Notepad.

➤ Open the plotter driver file in Notepad

1. With MicroStation still open, from the Windows Start button menu, choose Programs > Accessories > Notepad.

   Notepad opens.

2. From Notepad's File menu, choose Open.

   The Open dialog box opens.

3. Using controls in the Open dialog box, navigate to MicroStation's "Workspace\system\plotdrv" directory.

4. From the Open dialog box's Files of type option menu, choose All Files.

   By default, Notepad searches for files of type "*.txt."

   Changing this to All Files will display the plotter driver files which have the suffix ".plt."

5. In the list box, locate and select "ps650c.plt."

6. Click Open.

   The Open dialog box closes and the plotter driver file is opened in the Notepad window.

Before continuing, you should save this file with a different name so that the original file is left as is.

➤ Save the modified plotter driver file under a new name, in a separate directory

1. From Notepad's File menu, choose Save As.

   The Save As dialog box opens.

2. In the File name field, key in **ps650c_C_LS.plt.**

3. Use controls in the Save As dialog box to navigate to the "…\Workspace\standards\plotdrv" directory.

4. Click Save.

   The Save As dialog box closes and the filename in Notepad's title bar changes to the new name.

Look through the first part of this plotter driver file and find the line that begins `rotate = none`. Notice that instructions are included to the right of the default setting. These give you the other options that are available for this record.

➤ **Set the default rotation**

1. In the plotter driver file, for the `rotate` setting, change `none` to `cw`.

    This rotates the page 90° clockwise.

Next, you will change the default page size. When the plotter driver file is read into memory, the last sheet size listed is taken as the default. By moving the required sheet size to last in the list, you can change the default size that is used.

➤ **Set the default sheet size**

1. In Notepad, scroll through the plotter driver file to find the section headed "; ENGLISH resolution and SIZE records."

2. Place the pointer at the start of the line beginning: resolution (IN)=…

3. Press <Return> to create a blank line.

4. Highlight the line for the C sized sheet. That is, the line that ends with "/name=C."

5. From Notepad's Edit menu, choose Cut.

    The highlighted line is deleted, leaving a blank space where the C size record was.

6. Move the pointer down to the blank line created in step 3.

7. From Notepad's edit menu, choose Paste.

    The line is inserted.

8. Move the pointer back to the blank line where the C size sheet was originally.

9. Press <Delete> to remove the blank line.

```
ps650c_C_LS.plt - Notepad                                      _ □ ×
File  Edit  Search  Help

;   This configuration file contains the necessary records for English
;   and metric resolutions.  English units are used by default.  To use
;   the metric setting, comment out all lines in the ENGLISH section and
;   uncomment the RESOLUTION and SIZE lines in the METRIC section.  The
;   SIZE records define the plotting area of the page.

; ENGLISH resolution and SIZE records
size=( 6.9, 9.4)/num=4/off=(.8,.8)/name=A
size=( 9.4,15.4)/num=4/off=(.8,.8)/name=B
size=(20.4,32.4)/num=4/off=(.8,.8)/name=D
size=(32.4,42.4)/num=4/off=(.8,.8)/name=E
size=(15.4,20.4)/num=4/off=(.8,.8)/name=C
resolution(IN)=(0.003333333333333333,0.003333333333333333)  ;300DPI
```

10. From Notepad's File menu, choose Save.

Having created the new plotter driver file with the default settings that best suit you, you can select it for plotting.

➤ **Select the new plotter driver**

1. In the Plot Preview dialog box, click the Plotter Driver icon.
The Select Plotter Driver File dialog box opens.

2. From the list of Files, choose the new plotter driver file that you created in the previous exercise "ps650c_C_LS.plt."

3. Click OK.
The preview window updates showing the plot in a landscape orientation.

4. In the Plot Preview dialog box, click the Page Setup icon.
The Page Setup dialog box opens.
Notice that Rotate 90° is turned on and that the Page Size is C.

5. Click Cancel to close the Page Setup dialog box.

The changed plotter driver file provides the required default settings.

```
Page Setup
 Page Size:          ┌ Orientation: ┐     ┌──────────┐
 C              ▼                          │    OK    │
                                          └──────────┘
 Width:  20.400   IN.                     ┌──────────┐
 Height: 15.400   IN.                     │  Cancel  │
                                          └──────────┘
 ☑ Rotate 90°
```

Turning off a default plotter driver setting

Where you have a setting that you want to turn off, you simply place a semi-colon at the beginning of the line. For example, you may not want the plotter generated border, which is defined by the `border` record in the plotter driver file. In the plot preview window, the system generated plot border is represented by a rectangular shape enclosing the plotted area.
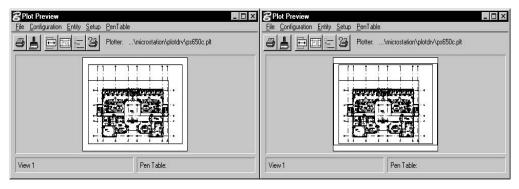
➤ **Turn off the border**

1. If necessary, re-open Notepad and select the plotter driver file "ps650c_C_LS.plt."

2. Locate the `border` record (10th. line of file).

3. Place a semi-colon (;) at the start of the line.

4. From Notepad's File menu, choose Save.

   Notice that there is no change in the plot preview window.

5. In the Plot Preview dialog box, click the Plotter Driver icon.

6. In the Select Plotter Driver dialog box, again select "ps650c_C_LS.plt."

   The plot preview window updates showing the proposed plot without a border.

Plot preview window showing the plot with a border (left) and without (right).

In the previous exercise, you disabled the border by adding a semi-colon at the beginning of the line or record. Everything to the right of a semi-colon is considered as a comment.

In a similar fashion, you can change the units of a plotter driver file from English to Metric. To do this, you comment out (add a semi-colon at the beginning of the line) all the English records and then delete the semi-colons from the beginning of the Metric records. This procedure is covered in a later tutorial, see "Changing plotting units from English to Metric" on page 7-6.

That concludes this tutorial.

5 Plotting Basics

This chapter discusses the steps required prior to creating a plot of your design file.

Introduction

Often, hard-copy plots or printouts are required of your MicroStation designs. These may range from simple "not to scale" plots for discussion purposes through to finished scale drawings.

With MicroStation you draw your designs at full-size. It is when it comes to plotting that you decide on the scale to be used. Plotting can be carried out from a view or from the contents of a fence. When a preliminary check plot is required, you can simply zoom in to or place a fence around the required area. You then plot this view or fenced area at the required scale. The scale can be set via the Plot Layout dialog box. For finished scale drawings, more planning is needed.

Plotting at the simplest level

At its most basic, preparing to plot consists of defining the part of the design file to plot, the type of plotter and the scale to plot. The following procedure describes the steps required to produce a plot of all or part of your design.

➤ **General procedure for plotting/printing a drawing**

1. Define the area you want to plot by:

   Placing a fence around the required area

   or

   Defining the part to be plotted with a view window.

2. From the File menu, choose Print/Plot.

   The Plot dialog box opens.

3. (Optional) From the Entity menu's View sub-menu, choose the view number (if a different view is to be plotted).

or

From the Entity menu, choose Fence.

(Fence will be the default selection if a fence is present prior to opening the Plot dialog box.)

4. (Optional) Use the Plot dialog box to select a different plotter driver and to make adjustments to the plot.

If you are using Windows, selecting "printer.plt" as your plotter will default to the system printer. You can define a default system printer using the Windows Print Manager. Alternatively, you can click the Page Setup icon, which opens the Print Setup dialog box. In this dialog box you can select another (Windows) printer as well as set the page size and rotation.

5. Plot to an attached printer or create a plotfile by:

In the Plot dialog box's icon bar, clicking the *Plot* icon.

or

From the File menu, choosing Plot.

✍ If "printer.plt" is selected as the plotter driver, plot output is sent to the system plotter driver. If another plotter driver is selected, plot output generally is sent to a plotfile, the name of which you can choose. This plotfile contains all the information required by the selected plotter to create the plot. An exception to this is when the default_outFile record in the plotter driver file designates a port name, such as "lpt1:" or printer name such as "\\printmanager\laserjet4." In these cases the plot will be sent directly to the port or printer.

Where a plotfile is created, this then can be copied to the plotter to create the plot.

Plotting to scale

Because MicroStation designs can be plotted at any scale, the one design can be used in a number of drawings. For example, the one design may be used in an overall plan at 1:100 and parts of it in other views at 1:10, 1:20 and 1:50. With manual drafting, each view would have to be drawn separately. With MicroStation, you need only draw the design once. It can be used any number of times, referenced to other files and plotted at various scales.

For this reason, planning is required to produce finished drawings with title blocks. Take text placement, for example. If you place text elements in the design, they too will be scaled along with the design elements.

With manual drafting, you scaled your design to fit the title block. MicroStation lets you use this same technique, via scaled reference files. Alternatively, you can scale your title block to fit the design.

Title block considerations

When working on a project, title blocks should be drawn in a separate design file and referenced when creating a drawing. For convenience, the title block design file should use the same working units as those used for the actual design. Having them in the same working units as the design simplifies the process of calculating the scale when creating the final drawings, as discussed in the following.

Using a 1:1 scale title block (recommended method)

With manual drafting, you have a full-size drawing sheet into which you place scaled views of your design. With MicroStation you can do the same thing, except that you draw the original design at full-size.

For this method, you leave the design geometry untouched. Using Drawing Composition, you create a sheet file or a sheet view into which you reference your standard title block (at full scale). Within the confines of this title block you then reference the design(s) at the required scale. All text and dimensioning is placed in the sheet file or view at full size.

➤ **General procedure to create a drawing with a 1:1 scale title block**

1. Using Drawing Composition, create a separate sheet file or sheet view.

2. Attach a full-size (1:1 scale) title block.

3. Into the title block, reference the required view(s) of your design at the necessary scale(s).

4. Place all text in the sheet file/view at its full-size.

Using this method, you place a fence around the title block and plot the design file at full-size. The title block and text will plot at their normal size while the design elements will plot to the scale

at which they were referenced to the title block. If you use this method to create your drawing files, it is very simple to use BatchPlot to plot sets of drawings in one step (refer to chapter 9, "Batch Printing/Plotting").

Scaling the title block to fit the design

An alternative method to that discussed above, is to scale a title block to surround the required part of the design. When you place text with this method, you have to calculate its size by applying the title block scale. For example, if the title block is 20 times its normal size, all text also would have to be 20 times its normal size. In such a case, when the final plot is created, you would use a scale of 1:20 to reduce the title block and text back to their normal size. At the same time, the design elements will be reduced by the same ratio.

➤ **General procedure to create a drawing with a scaled title block**

1. Prepare a view that shows the part of the design that you want to make into a drawing.

2. Reference a title block to the design at a scale that encloses the required area.

3. Place any text that is required for the finished drawing. Size of text should be scaled by the same ratio as that used for the title block.

With this method, you place a fence around the title block and plot the design file at a scale that returns the title block back to its normal size. The design elements and text are scaled by the same amount.

Introduction to the Plot dialog box

All plotting functions can be performed from the Plot dialog box. Using the options in this dialog box you can select a plotter and adjust various settings that affect plotting. Additionally, you can preview the plot.

You can define the area to be plotted by setting up a view to display the required area. Alternatively, you can place a fence around the area of interest. If a fence is present prior to opening the Plot dialog box, the fenced area is selected, by default, as the entity to be plotted.

➤ **To open the Plot dialog box**

1. From the File menu, choose Print/Plot.

or

Press <Ctrl + P>.

Plot dialog box.

All options for adjusting plotting settings are contained in the menu bar at the top of this dialog box. Commonly used options also can be selected via icons in the icon bar directly below the menu bar. The currently selected plotter driver is displayed in the field to the right of the buttons. Status bars at the bottom of the settings box display:

Left — the Entity being plotted (View, Fence or Configuration)

Right — the Pen Table being used (if any).

By default, the plot is maximized. That is, it is drawn to the largest scale that will fit on the selected paper size. This can be checked visually via the plot preview function.

➤ **To display a plot preview**
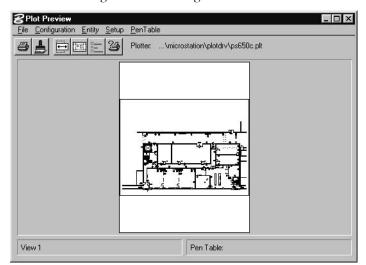
1. In the Plot dialog box's icon bar, click the Preview Refresh icon.

or

From the Plot dialog box's File menu, choose Preview.

The Plot dialog box expands to display the preview image. The Plot dialog box's title changes to Plot Preview.

Plot Preview dialog box.

To dismiss the preview window, you can shrink it down vertically until it disappears. Alternatively, you can reselect File > Print/Plot from the main menu bar to dismiss it.

Choosing the entity to plot

If a fence was active in the design file prior to opening the Plot dialog box, the plot area is taken to be the contents of this fence. If there is no fence present, the contents of the lowest numbered (displayed) view are used. You can change the entity that is to be plotted.

➤ **To select the entity to plot**

1. From the Plot dialog box's Entity menu, choose from the following:

Entity menu item	Used to plot the
View > 1…8	Contents of a view.
Fence	Fenced area.
Configuration	Area specified in the selected plot configuration file (see "Plot configuration files" on page 5-20).

The Plot Preview (if open) updates to display the selection.

Initially, the plot is calculated for the default sheet size and default orientation of the chosen plotter. You can change these and other settings as required. As you make changes, the Plot Preview window updates to display the result of the change. Prior to plotting, you can:

• Select a different plotter. See "Selecting a plotter" on page 5-8.

• Modify the page size and orientation of the plot. See "Setting the page size and orientation" on page 5-12.

• Position the plot on the sheet and set the plot scale. See "Plot layout criteria" on page 5-14.

• Add a plot border and description. See "Setting plot options" on page 5-10.

• Create a new plot configuration file or load an existing plot configuration file that defines the plotting settings. See "Plot configuration files" on page 5-20.

• Create a new pen table or load an existing pen table to resymbolize the plotted output. See "Pen tables" on page 11-2.

Preparing to create a plot

Prior to plotting your design, you first have to choose the type of plotter. Having selected the plotter, you then need to choose the page size and specify the layout of the plot on the selected page.

Page setup criteria

Choices for page sizes are determined by the type of plotter that you select. To set up the page for your plot you have to:

• Choose the plotting device (see "Selecting a plotter" on page 5-8).

• Set the plotting options (see "Setting plot options" on page 5-10).

• Set the paper size and orientation (see "Setting the page size and orientation" on page 5-12).

Selecting a plotter

MicroStation supplies driver files for most popular printers/plotters. These files are stored in the ".../Workspace/system/plotdrv" directory. One of these drivers "printer.plt" is specifically for use with a System printer. On a Windows system, you should use this driver to print to the Windows system printer. Where a number of networked printers are available, "printer.plt" by default prints to the default Windows printer. You can, however, select another of the networked printers (see "To select a networked plotter other than the Windows default" on page 5-10).
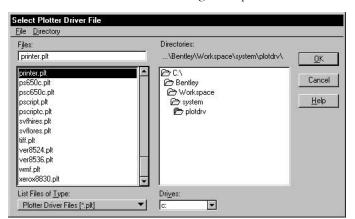
Another directory ".../Workspace/standards/plotdrv" is provided for saving your customized or frequently used plotter driver files. In particular, it is recommended that you store your customized plotter driver files in this directory. This minimizes the risk of losing your plotter driver files during a product reinstallation.

➤ **To select a printer/plotter**

1. In the Plot dialog box's icon bar, click the Plotter Driver icon

or

From the Plot dialog box's Setup menu, choose Driver.

The Select Plotter Driver File dialog box opens.

Select Plotter Driver File dialog box.

2. Select the new plotter driver.

3. Click OK.

The Select Plotter Driver File dialog box closes. If the Plot Preview is displayed, it updates to preview the plot from the new plotter.

Each plotter driver file is an ASCII text file that contains information relative to the plotter. This includes pages sizes (both English and Metric), the number of pens, default settings for rotation and borders. These default values may be altered using an ASCII text editor such as Notepad.

➤ **To select a networked plotter other than the Windows default**
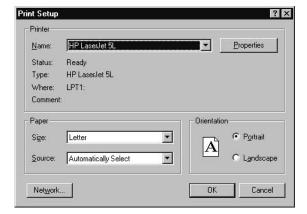
1. Follow the previous procedure selecting "printer.plt" as the plotter driver.

2. In the Plot dialog box's icon bar, click the Page Setup icon.

   or

   From the Plot dialog box's Setup menu, choose Page.

   The Print Setup dialog box opens. As well as settings for Paper size and Orientation, this dialog box has options for the selected (Windows) system printer/plotter.

Typical Print Setup dialog box for "printer.plt."

3. From the Printer Name option menu, choose the required printer.

4. Click OK.

   The Print Setup dialog box closes.

Setting plot options

If the entity to plot is a view or fence, the view attribute settings (Settings menu > View Attributes) determine most aspects of the plot's appearance. Check boxes in the Plot Options dialog box are dimmed (disabled) with the exception of Fence Boundary and Plot Border. These options let you include the Fence Boundary and/or a Plot Border with your plot.

When you select

• Fence Boundary — the plotted output includes the fence shape.

• Plot Border — the plotted output includes a border around the plot. Outside the border, a label can be placed giving

information such as the name of the design file and the time and date that the plot was produced. By default, the supplied plotter driver files have the variables "filename" and "time" included in the border record. These variables and other information that can appear in this label are configurable via the plotter driver file (see "border" on page B-2).

When you turn on Plot Border, you can add text in the Description field. This too appears in the label outside the plot boundary. You can include configuration variable references in this description. These are expanded in the plotted output. For example, if the configuration variable USER was defined as "John Smith" and you keyed in the description "User=$(USER)," the plotted output would expand the configuration variable so that the text "User=John Smith" appears in the label outside the plot boundary.

➤ **To add a border and description**

1. In the Plot dialog box's icon bar, click the Plot Options icon.

   or

   From the Plot dialog box's Setup menu, choose Options.

   The Plot Options dialog box opens.

2. Turn on Plot Border.

3. (Optional) In the Description field, key in a description (such as Office Plan).

4. Click OK.

The Plot Options dialog box closes.

c:/dgns/plan.dgn   Feb. 11, 1999   12:42:26 Office Plan

Example plot showing the border, file information and Description "Office Plan."

Setting the page size and orientation

Most plotter drivers have a choice of page sizes. For example, the "ps650c.plt" plotter driver file has page sizes A, B, C, D and E for the English configuration. For the Metric configuration, it has page sizes A0, A1, A2, A3 and A4. For a tutorial on changing the plotting units, see "Changing plotting units from English to Metric" on page 7-6. The orientation of the plot also may be set to zero rotation or rotated 90°.

✎ On some plotters, plots are not properly generated unless the page size specified is the same as the actual size of the paper loaded in the plotter.

✎ On some systems, when "printer.plt" is selected, the Printer Setup dialog box lets you connect to other installed printers.
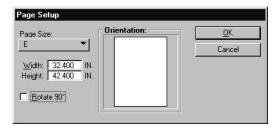
➤ **To define the page setup for the plot**

1. In the Plot dialog box's icon bar, click the Page Setup icon.

or

From the Plot dialog box's Setup menu, choose Page.

The Page Setup dialog box opens.

Typical Page Setup dialog box for plotters other than "printer.plt."

If "printer.plt" is the selected plotter driver file, then the Print Setup dialog box opens. This dialog box has options for the selected (Windows) system printer/plotter, which differ from those in the Page Setup dialog box.

Typical Print Setup dialog box for "printer.plt."

2. (Optional) From the Page Size option menu, choose the new page size.

or

(Optional) In the Width and/or Height fields, key in new values.

3. (Optional) Turn on Rotate 90° to change the orientation of the page.

4. Click OK.

The Page Setup (or Print Setup) dialog box closes. If the Plot Preview is displayed, it updates to preview the plot for the new page size and/or rotation.

Plot layout criteria

Prior to plotting your design, you can set the plot dimensions or scale to what is required. You can change the Plot Width, Plot Height or Scale. These settings are interlocked to preserve the aspect ratio of the plot area. Changing one setting results in changes to the others. You cannot, however, set the plot Height, Width or Scale settings to a figure that would place part of the plot outside the area of the selected page.

When a plot is maximized, it is drawn to the largest scale that will fit on the selected paper size. That is, either its width or height is scaled to match that of the selected page with the aspect ratio determining the remaining dimension. You can set the scale to be a percentage of this normal (page) size, or as a ratio of the master units to the plotter units (plotter units are specified in the plotter driver file's RESOLUTION record). When a plot takes up only a portion of the selected page size, you can position the plot on the page by adjusting the margins. By default, Center to Page is turned on. This ensures that the plot always is centered in the selected page. If you specify a value for either margin, Center to Page is turned off.

Using settings in the Plot Layout dialog box you can:

- Set the dimensions of the plot (see "Setting the plot Height or Width" on page 5-14).

- Set the scale of the plot relative to the page size (see "Setting plot scale relative to the page size" on page 5-16).

- Set the scale of the plot as a ratio of working units to plotter unit (see "Setting the exact scale of a plot" on page 5-15).

- Set the Left and Bottom margins to position the plot on the selected page (see "Setting the Left and/or Bottom margins" on page 5-17).

- Maximize the plot or center it on the page (see "Center to Page and Maximize settings" on page 5-18).

Setting the plot Height or Width
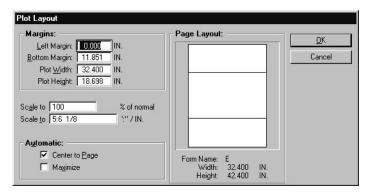
Within the limits of the selected page size and, if specified, the Left and Bottom margins, you can set the Height or the Width of

your plot. At all times the aspect ratio of the plot is maintained. Setting the Plot Height automatically sets the Plot Width and vice-versa.

➤ **To set the plot height or width**
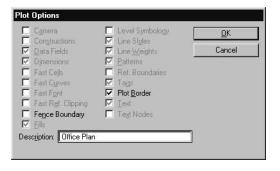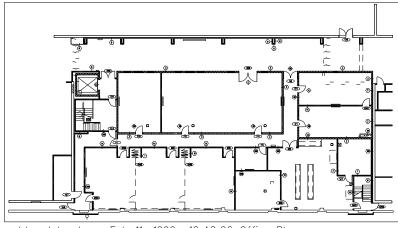
1. In the Plot dialog box's icon bar, click the Plot Layout icon.

    or

    From the Plot dialog box's Setup menu, choose Layout.

    The Plot Layout dialog box opens. In its Page Layout section, the black rectangle represents the selected page for the chosen plotter. The blue rectangle inside it represents the plot area. If Fence is the chosen plotting entity, the blue rectangle represents the smallest rectangular area that encloses the fenced area.

2. In the Plot Width or Plot Height field, key in the required value.

    The remaining values update to reflect the new setting. The blue rectangle also updates to show its size relative to the sheet size.

3. Click OK.

    The Plot Layout dialog box closes.

Setting the exact scale of a plot

Each plotter driver has its own plotting units. By default, the plotter driver files supplied with MicroStation are in English units (IN). You can edit the plotter driver file to change the units to Metric (MM) (see "Changing plotting units from English to Metric" on page 7-6).

In either case, you can specify the scale of your plot as a ratio of MU:SU to Plotter Units. Consider, for example, your design file

having Working Units of Feet (MU) and Inches (SU), while your plotter's units are Inches.

To create a 15 feet to the inch scale plot, you would enter 15 in the *Scale to ft:in/IN* field. That is, the plot would be 15 ft per IN.

To create a 1:20 scale plot, you would enter 0:20 in the *Scale to ft:in/IN* field. That is, the plot would be 0ft.-20in. per IN.

✍ You cannot set a scale that would place part of the plot outside the limits of the selected page size.

➤ **To set the exact scale of the plot**

1. In the Plot dialog box's icon bar, click the Plot Layout icon.

   or

   From the Plot dialog box's Setup menu, choose Layout.

   The Plot Layout dialog box opens. In its Page Layout section, the black rectangle represents the selected page for the chosen plotter. The blue rectangle inside it represents the plot area. If Fence is the chosen plotting entity, the blue rectangle represents the smallest rectangular area that encloses the fenced area.

Plot Layout		
Margins:	**Page Layout:**	OK
Left Margin: 0.000 IN.		Cancel
Bottom Margin: 11.851 IN.		
Plot Width: 32.400 IN.		
Plot Height: 18.698 IN.		
Scale to 100 % of normal		
Scale to 5:6 1/8 '." / IN.		
Automatic:	Form Name: E	
☑ Center to Page	Width: 32.400 IN.	
☐ Maximize	Height: 42.400 IN.	

2. In the Scale to MU:SU/Plotter Units field, key in the required value.

3. Click OK.

   The Plot Layout dialog box closes.

Setting plot scale relative to the page size

By default, MicroStation maximizes the plot on the selected page. You can then scale the plot relative to the page. This can be used for presentation purposes, for example, where a true scale plot is not required.

➤ **To set the scale of the plot relative to the page**

1. In the Plot dialog box's icon bar, click the Plot Layout icon.

or

From the Plot dialog box's Setup menu, choose Layout.

The Plot Layout dialog box opens. In its Page Layout section, the black rectangle represents the selected page for the chosen plotter. The blue rectangle inside it represents the plot area. If Fence is the chosen plotting entity, the blue rectangle represents the smallest rectangular area that encloses the fenced area.

2. In the Scale to % of normal field, key in the required value.

The remaining values update to reflect the new setting. The blue rectangle also updates to show its size relative to the sheet size.

3. Click OK.

The Plot Layout dialog box closes.

Setting the Left and/or Bottom margins

Where the plot area is smaller than the page on which it is being plotted, you can use the Left Margin and Bottom Margin settings to position the plot on the selected page.

➤ **To position the plot on the page**
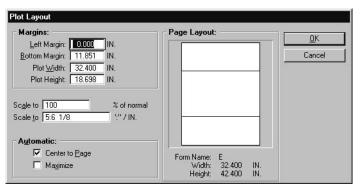
1. In the Plot dialog box's icon bar, click the Plot Layout icon.

or

From the Plot dialog box's Setup menu, choose Layout.

The Plot Layout dialog box opens. In its Page Layout section, the black rectangle represents the selected page for the chosen plotter. The blue rectangle inside it represents the plot area. If Fence is the chosen plotting entity, the blue rectangle

represents the smallest rectangular area that encloses the fenced area.

By default, the plotting entity is centered on the page.
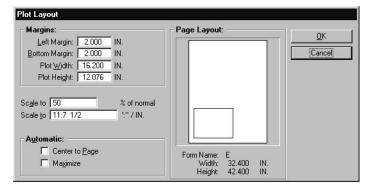
2. In the Left Margin and/or Bottom Margin fields, key in the new values.

As the new values are entered and you <Tab> between the fields, the blue rectangle in the Page Layout section of the dialog box updates to display the new location of the plot in the selected page.

When a value is entered for either margin, Center to Page is turned off.

3. Click OK.

The Plot Layout dialog box closes.

Plot Layout dialog box showing plot area relocated on sheet.

Center to Page and Maximize settings

By default, when you open the Plot dialog box, the plot is maximized. That is, it is drawn to the largest scale that will fit on the selected paper size. By default, the plot is centered on the page. When you make adjustments to the margin settings, Center

to Page turns off. At any time, you can turn on Center to Page or Maximize to center the plot or maximize it on the page. When Maximize is turned on, Center to Page also turns on (automatically).

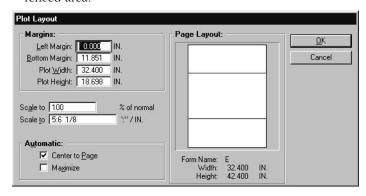➤ **To center the plot on the page**
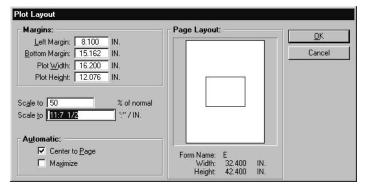
1. In the Plot dialog box's icon bar, click the Plot Layout icon.

    or

    From the Plot dialog box's Setup menu, choose Layout.

    The Plot Layout dialog box opens. In its Page Layout section, the black rectangle represents the selected page for the chosen plotter. The blue rectangle inside it represents the plot area. If Fence is the chosen plotting entity, the blue rectangle represents the smallest rectangular area that encloses the fenced area.

2. Turn on Center to Page.

    The blue rectangle updates to be central on the page in the Page Layout group box. The Left Margin and Right Margin settings update to reflect the new position.

Plot Layout dialog box, showing the plot centered in the page.
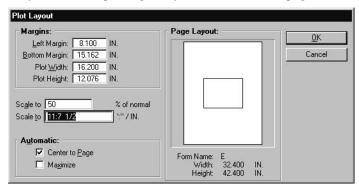

➤ **To maximize the plot**
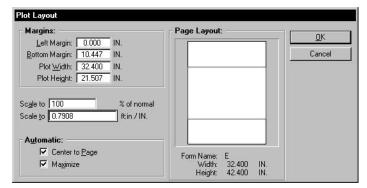
1. In the Plot dialog box's icon bar, click the Plot Layout icon.

    or

    From the Plot dialog box's Setup menu, choose Layout.

    The Plot Layout dialog box opens. In its Page Layout section, the black rectangle represents the selected page for the chosen plotter. The blue rectangle inside it represents the plot area. If Fence is the chosen plotting entity, the blue rectangle represents the smallest rectangular area that encloses the fenced area.

2. Turn on Maximize.

By default, Center to Page also is turned on.

Plot Layout dialog box showing the plot maximized on the sheet.

Plot Layout

Margins:
Left Margin: 0.000 IN.
Bottom Margin: 10.447 IN.
Plot Width: 32.400 IN.
Plot Height: 21.507 IN.

Scale to 100   % of normal
Scale to 0.7908   ft:in / IN.

Automatic:
☑ Center to Page
☑ Maximize

Page Layout:

Form Name: E
Width: 32.400 IN.
Height: 42.400 IN.

OK
Cancel

Plot configuration files

It is not necessary to go through the complete setup procedure each time that you require a plot from a design file. You can create a saved **plot configuration file**. These specify the design file-specific information required to recreate a particular plot. Using plot configuration files is a way to streamline repetitive plotting tasks.

A plot configuration file consists of the following entities and settings:

• Specification of the plotted area of the design.

• View attributes as set in the Plot Options dialog box.

• Levels of the master file displayed in the plotted view (reference file levels are not stored).

• Page size, margins and scale.

• Pen table if attached.

You can use plot configuration files on design files other than the one used to create them with one proviso. The plot configuration file must have been created from a design file having the same working units as that of the active file.

By default, plot configuration files are given the suffix ".ini." They are ASCII text files that, by default, are saved in MicroStation's "Workspace\system\data" directory.

➤ **To create a plot configuration file**

1. Use the controls in the Plot dialog box to distinguish the entity to plot and adjust the plot setup as desired.

2. From the dialog box's Configuration menu, choose New.

   The Plot Configuration dialog box opens.

3. In the Files field, key in the new plot configuration file name. By default, the extension ".ini" is added to the file name.

4. Click OK.

➤ **To modify the active plot configuration file**

1. Use the controls in the Plot dialog box to modify the entity to plot and/or adjust other plot settings as desired.

2. From the dialog box's Configuration menu, choose Save.

   The new settings are saved in the active plot configuration file.

When one or more plot configuration files have been created for a design file, you can use these to create plots in the future. This saves having to remember which levels are displayed, the position of the fence or the size of the view, the type of plotter and so on.

➤ **To open an existing plot configuration file**

1. From the Plot dialog box's Configuration menu, choose Open.

   The Plot Configuration File dialog box opens.

2. Select the required plot configuration file.

3. Click OK.

   The Plot Configuration File dialog box closes and the settings for the selected configuration file are applied to the Plot dialog box.

6 Tutorial: Basic Plotting

This tutorial takes you through the basic steps for creating plots of your designs.

Overview

In the following exercises, you will learn how to set up to plot the contents of a view and a fence to the system printer. During the exercises, you will use the Preview window to study how the various settings affect the plot.

Preparation

For the purposes of this tutorial, the system printer will be used. That is, it is assumed that a printer is connected to the system or is accessible via a network and that it has been configured as the default Windows printer.

As these plots will be similar to check plots, we will turn on the border so that the name of the design file and the time of plotting is noted on the plot.

For the exercise, we will use the sample design file "floor.dgn."

➤ **Prepare for the exercise**

1. Open the design file "floor.dgn," which is located in the "…\Workspace\projects\tutorials\intro\dgn" directory.

2. Use the *Fit View* view control to fit View 3.

   View 3 displays the overall floor plan of an office.

3. From the File menu, choose Print/Plot.

   The Plot dialog box opens.

4. In the Plot dialog box, click the Plotter Driver icon.

   The Select Plotter Driver File dialog box opens.

5. In the Files list box, select "printer.plt."

This is the driver for the system printer.

6. Click OK.

The dialog box closes and the name of the system printer plus the default page size appears in the title bar of the Plot dialog box.

Plotting the contents of a view
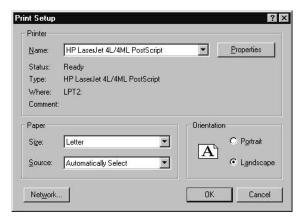
Page sizes vary with the type of plotter being used. For this exercise, we are assuming a plotter that has a paper size "Letter" (8.5 in. x 11 in.). If your plotter does not have this size, just select the nearest to that size.

➤ **Set the page size and orientation**

1. In the Plot Preview dialog box, click the Page Setup icon.

The Print Setup dialog box opens. Because the system printer is being used, the printer's setup dialog box is used for the sheet setup.

2. From the Paper Size options menu, choose Letter.

3. If necessary, turn on Landscape.

4. Click OK.

Print Setup dialog box for the system printer.

With the page setup complete, we can look at the plot preview to check that the plot is set up correctly.

➤ Preview the plot

1. In the Plot dialog box, click the Preview Refresh icon to display the plot preview.

    Currently, the preview window displays the contents of View 1, the lowest numbered view.

2. From the Entity menu's View sub-menu select 3 to set the plot contents to that of View 3.

    The preview window updates to display the contents of View 3.

Next, turn on the border so that the design file's name and when it was plotted is recorded. As well, you can add a comment — for example, to indicate the scale of the plot.

➤ Turn on the border

1. In the Plot dialog box's icon bar, click the Plot Options icon. The Plot Options dialog box opens.

2. Turn on Plot Border.

3. In the Description field, key in **Scale 1″=10′**.

4. Click OK.

    The Plot Options dialog box closes.

Currently, the plot is maximized (fitted) to the selected page size. We will now set the scale to 10 feet per inch. For this part of the exercise, it is assumed that your plotter's units are in inches.
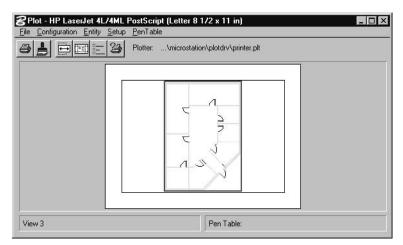
➤ Set the scale and submit the plot

1. In the Plot Preview dialog box, click the Plot Layout icon. The Plot Layout dialog box opens.

2. In the `Scale to ':"/IN` field, key in **10**.

3. Click OK.

The Plot Preview window updates to display the scaled plot information.

Plot Preview window displaying the plot information.

4. In the Plot dialog box, click the Plot icon to submit the plot directly to the printer.
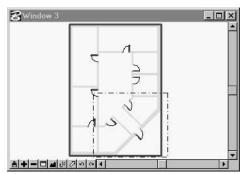
Plotting the contents of a fence

Following on from the previous exercise, next you will place a fence around the entrance area. This will be used to create a larger scale plot of that part of the design.

➤ **Create a larger scale plot of the entrance area**

1. In View 3, use the *Place Fence* tool to place a fence around the entrance area (lower right) of the floor plan.

2. From the Plot dialog box's Entity menu, choose Fence.

The Preview Refresh window updates to display the fence contents.

3. In the Plot Preview dialog box, click the Plot Layout icon.

The Plot Layout dialog box opens.

4. In the `Scale to ':"/IN` field, key in **5**.

5. Click OK.

The Plot Preview window updates.

All that remains is to change the comment for the border to reflect the new scale for this plot.

➤ **Change the border comment and submit the plot**

1. In the Plot dialog box's icon bar, click the Plot Options icon.

The Plot Options dialog box opens.

2. In the Description field, key in **Scale 1"=5'**.

3. Click OK.

The Plot Options dialog box closes.

4. In the Plot dialog box's icon bar, click the *Plot* icon to submit the plot directly to the printer.

Creating a plotfile for later submission to a plotter

It is not necessary to submit plots directly to a printer. You can create plotfiles for a printer attached to your system or for any other printer on the network. These are saved to disk, letting you submit them to the plotter at another time.

So that the plot is not sent directly to the system printer, you should first change the plotter driver file from "printer.plt" to that for the particular plotter that you want to use.

➤ **Select the required plotter driver file**

1. In the Plot dialog box, click the Plotter Driver icon.

   The Select Plotter Driver File dialog box opens.

2. In the Files list box, select the plotter driver file for your printer.

3. Click OK.

You now can create a plotfile that is saved to disk.

➤ **Create a plotfile of the contents of View 3**

1. From the Entity menu's View menu, choose View 3.

2. In the Plot Preview dialog box, click the Plot Layout icon.

   The Plot Layout dialog box opens.

3. In the Scale to ':"/IN field, key in **10**.

4. Click OK.

5. In the Plot dialog box's icon bar, click the Plot icon.

   The Save Plot As dialog box opens. By default, the plotfile is given the name of the design file, with an extension ".000." If a plotfile of this name exists, the extension is incremented by 1 (to ".001").

6. Use the controls in the Save Plot As dialog box to select the directory in which you want to store your plotfiles (you will need to remember this directory so that you can find the plotfiles later).

7. Click OK.

   The plotfile is created. The message <Finished Creating Plot> appears in the status bar when the process is complete.

Similarly, further plotfiles can be created.

➤ **Create a second plotfile (of the entrance area)**

1. In View 1, use the *Window Area* view control to zoom in on the entrance area of the office layout.

This is the area around which you placed a fence earlier.

2. In the Plot Preview dialog box, click the Plot Layout icon.

The Plot Layout dialog box opens.

3. In the `Scale to ':"/IN` field, key in **5**.

4. Click OK.

5. In the Plot dialog box's icon bar, click the Plot icon.

The Save Plot As dialog box opens. Notice that the directory used in the previous plot now is the default. Also note that the extension for the plotfile is incremented by 1 from the previous plot.

6. Use the controls in the Save Plot As dialog box to select the directory in which you want to store your plotfiles (you will need to remember this directory so that you can find the plotfiles later).

7. Click OK.

The plotfile is created. The message <Finished Creating Plot> appears in the status bar when the process is complete.

Submitting plotfiles to a plotter

In the previous exercises you created plotfiles for a particular plotter or printer. You can produce hardcopy plots from these plotfiles by copying them to the printer. This is done from a DOS session window.

Because of the diversity of setups, we will make assumptions for the following exercise. For the following procedure, we will assume that you have saved the plotfiles in a directory "c:\plots" and that your plotter/printer is on port "\\printman\hp7475" (you should use your port if you are following the exercise). You can copy a single plotfile to the printer or use wildcard entries to submit a number of plots in a single command line.

➤ **Copy the first plotfile to the printer/plotter (Windows)**

1. From the Start button's Programs menu, choose Command Prompt (Windows NT) or MS-DOS Prompt (Windows 95).

The Command Prompt/MS-DOS Prompt window opens.

2. In the system window, key in **cd\plots.**

   This changes the directory to that where the plotfiles were saved. Here you can use the **dir** key-in to view the files in that directory.

3. In the system window, key in:

   copy /b office.001 \\printman\hp7475

   This key-in sends the first plot to the printer (substitute your port for \\printman\hp7475).

Where you have a number of plots from the same design file, it is convenient to use a wildcard entry to copy all plotfiles to the printer with one command.

➤ **Copy a number of plotfiles to the printer/plotfile (Windows)**

1. In the system window, key in:

   copy /b office.* \\printman\hp7475.

Using this method lets you quickly create all your plotfiles and then copy them to the plotter at a convenient time. Using the default naming system where the design file's name has a numeric extension added to it lets you submit all the plotfiles with a single key-in.

7

Tutorial: Plotting to Scale

This tutorial leads you through the steps required to create scale drawing plots.

Preparation

For the following exercise you will work with the sample design file "plot2sca.dgn." This 3D design file contains three 2D views of a simple bracket.

During the exercise, you will use the Plot Preview window to examine the results of changes made to the plotter settings. The delivered plotter configuration file `ps650c.plt` will be used. This is a mono postscript configuration file for a large format plotter.
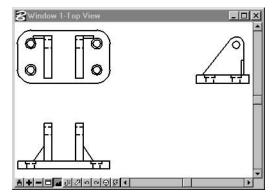
➤ **Prepare for the exercise**

1. Open the design file "plot2sca.dgn," which is located in the "Bentley\Workspace\projects\tutorials\intro\dgn" directory.

   This design file has been saved with View 1 open, a Top view, displaying three (2D) elevations of the bracket that will comprise the drawing.

2. Use the *Fit View* view control to fit View 1.

3. From the File menu, choose Print/Plot.

   or

   Press <Ctrl+P>.

   The Plot dialog box opens.

4. In the Plot dialog box, click the Plotter Driver icon.

   The Select Plotter Driver File dialog box opens.

5. In the Files list box, select `ps650c.plt`.

   The file name appears in the field above the list box.

   Note that `ps650c.plt` is the mono postscript driver, while `psc650c.plt` is for color postscript.

6. Click OK.

   The Select Plotter Driver File dialog box closes.

   In the Plot dialog box, the selected plotter's name appears in the Plotter field to the right of the icon bar.

Plotting a Metric units drawing on an English units plotter

When setting the scale of a Metric drawing to be plotted on a plotter using English units, you have to make allowance for the conversion factor.

Plotting contents of a view
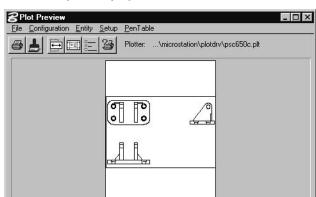
For this exercise, you will plot the contents of View 1.

➤ **Prepare to plot the contents of a view**

1. In the Plot dialog box, click the Preview Refresh icon.

   The preview window opens and the dialog box's name changes to Plot Preview. The preview window displays the contents of the open view that are to be plotted.

   By default, the plot is maximized to fit the default sheet size for the selected plotter. For the plotter selected, this is an E size sheet.

For this exercise, we will use a C size sheet in a landscape orientation (rotated 90°).

➤ **Select the sheet size and orientation**

1. In the Plot Preview dialog box, click the Page Setup icon. The Page Setup dialog box opens.
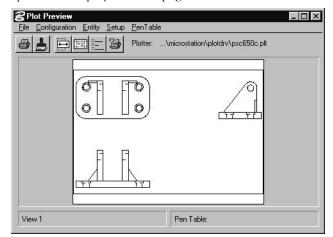
2. From the Page Size options menu, choose C.

3. Turn on Rotate 90°.

4. Click OK.

The Page Setup dialog box closes. The Plot Preview window updates to display the new page size and orientation.

Plot Preview dialog box showing the new page size and orientation.

With the page size and orientation selected, you will now determine the scale of the plot. In this case, the design has been created in Metric units with the Master Units set as mm (millimeters). As delivered, the plotter configuration files are in English units. The scale is defined in terms of Master Units per Plotter Unit. To calculate the scale, you will have to take into account that there are 25.4 mm per inch. To produce a 1:1 scale plot, you will set the scale to 25.4mm/IN.
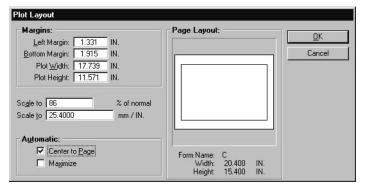
➤ **Set the scale to create a 1:1 plot**

1. In the Plot Preview dialog box, click the Plot Layout icon.

The Plot Layout dialog box opens. In the Plot Layout section, the blue rectangle represents the current size of your plot.

2. In the Scale to mm/IN field, key in **25.4**, followed by <TAB>.

In the Plot Layout section, the blue rectangle shrinks to show the relative size of our plot to the selected sheet size.

Plot Layout dialog box after setting the scale to 25.4mm/IN (that is, 1:1).

You now should set the options prior to plotting. For example, currently the plot preview displays a border surrounding the design (if yours does not have the border, then it is probably already turned off). When plotted, this border also will be plotted along with the name of the file and the time and date of the plot. Via the plot options, you can turn off this border. You can leave it turned on and you can add a brief description to be printed following the default information.

➤ **Turn off the default border**

1. In the Plot Preview dialog box, click the Plot Options icon.

The Plot Options dialog box opens.

2. If necessary, turn off Plot Border.

3. Click OK.

The Plot Options dialog box closes. The plot preview window updates to show the plot without the default border.

If your plotter is a mono postscript plotter compatible with the plotter driver file that we selected at the outset, you would now simply click the Plot icon. You could either create a plotfile or where the plotter is the system printer, you could send the plot directly to the plotter (see Chapter 8, Submitting Plots to a Printer/Plotter).

Changing plotting units from English to Metric

By default the plotter driver files delivered with MicroStation are configured for English units. However, they also contain the Metric configuration information.

In the previous exercise, the design was in Metric units while the plotter configuration was in English units. In the following exercise, you will create a metric plotter driver from the plotfile "ps650c.plt," the driver used in the previous exercise. To do this, you will need an ASCII text editor such as Notepad, which comes with Windows products. Typically, it is located in the Accessories folder and can be started by selecting Start > Programs > Accessories > Notepad.

✐ When you use "printer.plt" as your plotter driver file, a system printer is used. In these cases the measurement system used is defined by Windows' Control Panel - Regional Settings, in the Numbers tab. Options are US or Metric.

➤ **Open the plotter driver file in Notepad**

1. From Notepad's File menu, choose Open.

The Open dialog box opens.

2. Use the controls in the Open dialog box to select the directory "…\Workspace\system\plotdrv."

3. From the Files of type option menu in the Open dialog box, choose All Files.

All files in the selected directory display.

4. From the list of files, select "ps650c.plt."

Be careful not to select "psc650c.plt" by mistake.

5. Click Open.

The selected plotter driver file opens in Notepad.

As you will see, by scrolling through this file, there are instructions for making changes to the various settings. In particular, look at the section containing the English and Metric resolution and SIZE records.

Above this section is a brief description of the settings and how to change from English to Metric, as follows:

```
This configuration file contains the necessary records
for English and metric resolutions. English units are
used by default. To use the metric setting, comment out
all lines in the ENGLISH section and uncomment the
```

```
RESOLUTION and SIZE lines in the METRIC section. The SIZE
records define the plotting area of the page.
```

All lines beginning with a semi-colon (;) are "commented out." That is, the plotting process ignores these lines. To comment out the English lines, you simply place a semi-colon at the beginning of each line. To activate the Metric information, you remove the semi-colons from the size and resolution records. When you do this, it is a good idea to save the file under a different name retaining the original file "as is."

➤ **Change the plotters units to Metric**

1. Before each of the English resolution and size records (lines) place a semi-colon.

2. In the Metric section, delete the semi-colons preceding each line for the size and resolution records.

3. From Notepad's File menu, choose Save As.

   The Save As dialog box opens.

4. Use controls in the Save As dialog box to navigate to the "…Workspace\standards\plotdrv" directory.

   It is recommended, for security, that any customized plotter driver files are saved in this directory. This reduces the chance of your customized files being deleted during a product reinstallation, for example.

5. In the File name field, key in **ps650c_m.plt**.

   The "m" signifies metric units.

6. Click Save.

7. Close Notepad.

Notepad window showing the English and Metric records after editing.

With the new metric plotter driver file you will now create a drawing from the various views of the model. You will assemble them within an A3 title block, ready for plotting.

Preparing a drawing ready to plot

In preparing a drawing of the bracket model, you will use another MicroStation feature — Drawing Composition.

First, you will place an A4 title block at full scale and then place the drawing components within the title block.

For this exercise, you will reference the A4 title block contained in the design file "A4Title.dgn," which has the same working units as those of the design geometry's design file.

✍ Having a title block prepared with the same design file units simplifies the task of determining scales.

➤ Create a sheet view with a title block

1. If necessary, open the design file "plot2sca.dgn."

2. From the File menu, choose Drawing Composition.

The Drawing Composition dialog box opens.
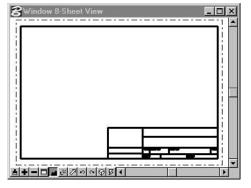
3. If necessary, from the Sheet Location option menu, choose Sheet View.

4. In the Drawing Composition dialog box select Tools > Open Sheet View > 8.

   View 8 opens, with the title Sheet View.

5. In the Drawing Composition dialog box select Tools > Attach Border > Coincident

   The Attach Border File dialog box opens.

6. Use the controls in the dialog box to select the file "A4Title.dgn" in the "…\Workspace\projects\examples\intro\dgn" directory.

7. Click OK.

   The dialog box closes and you are prompted to "Identify Drawing View."

8. Enter a data point in View 8 (the sheet view).

9. Use the *Fit View* view control to fit View 8.

   View 8 displays the title block.

View 8 (sheet view) displaying the A3 title block.

With the title block in place, you are ready now to place the views of the model. For this exercise, it is the one view.

➤ **Place the drawing components**
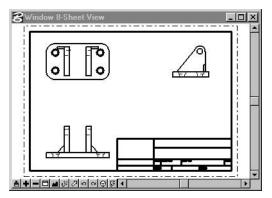
1. Use the *Zoom Out* view control to zoom out once in the sheet view (Zoom Ratio 2.0).

2. In the Drawing Composition dialog box select Tools > Attach Standard > Top.

3. In the sheet view, move the pointer to the center of the title block but do not enter a data point. Notice that the rectangle

representing the area in which the drawn elements are located is bigger than the title block.

4. In the Drawing Composition dialog box's Scale (Sheet : Model) model field (the right one), key in **2** followed by a <Tab>.

 The rectangle decreases in size. The scale for Sheet : Model will be 1:2. The model now is half-scale. The outer rectangle represents the Margin (%), which allows for placement of dimensions.

5. In the center of the title block, enter a data point to place the reference attachment.

Sheet view after placing the drawing component.

To complete the drawing, you would add text and dimensioning. Because the title block has been referenced at full-scale, the plot will be produced at a scale of 1:1. Therefore text sizes also can be full-size. That is, if you want 5mm text in the plot, you simply set the text size to 5mm.

For the exercise, you will use the metric version of the `ps650c.plt` plotter driver that you created in the earlier exercise.

➤ **Select the plotter**

1. From the File menu, choose Print/Plot.

 or

 Press <Ctrl+P>.

 The Plot dialog box opens.

2. In the Plot dialog box, click the Plotter Driver icon.

 The Select Plotter Driver File dialog box opens.

3. Use controls in the dialog box to navigate to the "…Workspace\standards\plotdrv" directory.

4. In the Files list box, select `ps650c_m.plt`.

5. Click OK.

The Select Plotter Driver File dialog box closes.

We are using an A4 title block, which has overall dimensions of 297mm by 210mm. These are represented by the dashed line outside the heavier solid line of the title block. Printers/plotters generally do not print to the very outer edge of pre-cut sheets, so some allowance has to be made for margins. In this case, the solid line of the title block has been drawn to dimensions that can be plotted on the selected plotter. It is this rectangle that we will use to place the fence defining the plotting area.

➤ **Place the fence to define the plot area**

1. Use the *Place Fence* tool to place a fence on the heavy solid line of the title block.

2. In the Plot dialog box, from the Entity menu, choose Fence.

3. If necessary, click the Preview Refresh icon to display the plot preview.
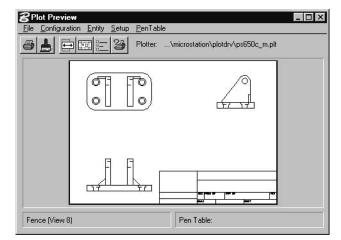
➤ **Select the page size and orientation**

1. In the Plot dialog box, click the Page Setup icon.

The Page Setup dialog box opens.

2. From the Page Size option menu, choose A4.

3. Turn on Rotate 90°.

4. Click OK.

The Plot Preview window updates to display the new page size and orientation.

Plot Preview dialog box after selecting the page size and orientation.

➤ **Set the scale for the plot**

1. In the Plot Preview dialog box, click the Plot Layout icon.

The Plot Layout dialog box opens. Notice that the Scale to setting is shown as "mm/MM" not "mm/IN" because the new plotter driver file uses metric units.

2. In the Scale to mm/MM field, key in **1.0**.

3. Click OK.

The Plot Layout dialog box closes.

This setup plots the design at a 1:1 scale so that the title block is produced at its correct size. The design elements were referenced in to the area of the title block at 2:1 scale and will be plotted at that scale.

With the drawing set up, you are ready to click the Plot icon and either create a plotfile or plot directly to an attached printer/plotter. For information on these processes (see Chapter 8, Submitting Plots to a Printer/Plotter).

8 Submitting Plots to a Printer/Plotter

This chapter discusses the steps for submitting plots to your printer/plotter.

Introduction

You can produce plots of your whole design or part of it at any time. You can create plotfiles on disk and submit them to a printer/plotter. You also can send plots directly to a printer/plotter.

Where plotfiles are used, rather than plotting directly to the plotter, you first have to create the plotfile.

Creating a plotfile from the Plot dialog box

You can create a plotfile from the Plot dialog box. Additionally, you can make adjustments to the various plotting settings prior to creating the plotfile. If a fence is present in the design file prior to opening the Plot dialog box, this becomes the default plotting entity. Where no fence is present, the lowest numbered (open) view window becomes the default plotting entity. You can change the plotting entity via the Plot dialog box.

➤ **To create a plotfile from the dialog box**

1. From the File menu, choose Print/Plot.
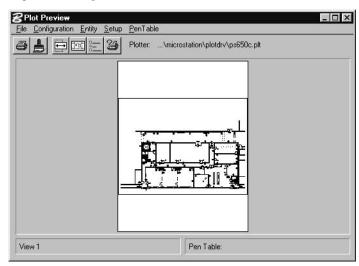
or

Press <Ctrl+P>.

The Plot dialog box opens. By default, the plot is **maximized**: it is drawn to the largest scale that will fit on the selected paper size.

2. (Optional) To display a plot preview image, click the Preview Refresh icon in the dialog box's icon bar.

Clicking the icon expands the dialog box to reveal the preview image.

To dismiss the preview image, shrink the preview window (vertically) until it disappears or reselect File > Print/Plot in the main menu bar.

3. (Optional) Select a different plotter driver file. See "Selecting a plotter" on page 5-8. If the system plotter driver file "printer.plt" is selected, plot output is sent to the system print manager. If not, plot output is sent to a plotfile that you name later.

4. (Optional) Select a **plot configuration file**, which is a previously saved plotting setup. See "Plot configuration files" on page 5-20.

   or

   Define the entity to plot. See "Choosing the entity to plot" on page 5-6.

   Changing the plot setup, either manually or by selecting a plot configuration file, updates the preview image. To manually update the preview image, click the Preview Refresh icon in the dialog box's icon bar.

5. (Optional) You can do any or all of the following:

Action	For more information, see
Adjust the Page Setup to: • change the paper size the design is to be plotted on. • rotate the plot 90° in the printable area.	"Setting the page size and orientation" on page 5-12
Adjust the Plot Layout to: • change the scale or plot margins. • position the plot on the sheet.	"Plot layout criteria" on page 5-14
Adjust the appearance of the plot, add a border and/or comment.	"Setting plot options" on page 5-10
Select a pen table for plot resymbolization.	"Pen tables" on page 11-2

6. In the Plot dialog box's icon bar, click the Plot icon.

   or

   From the Plot dialog box's File menu, choose Plot.

   If the plotter driver file "printer.plt" is selected, plot output is sent to the system print manager.

   If another plotter driver file is selected and its `default_outfile` record is not set to a port or other printer, then the Save Plot As dialog box opens and continue to step 7.

7. To name, create and save the plotfile, use the Save Plot As dialog box just as you would the Save Design As dialog box. The default plotfile extension is ".000" unless otherwise specified in a `DEFAULT_EXTENSION` record in the plotter driver file. If a numeric default extension is used, such as ".000," the extension proposed in the dialog box automatically increments with the creation of each additional plotfile having the same file name (for example, "bracket.000, bracket.001, bracket.002…).

✍ To open the Plot dialog box in preview mode, key in **PREVIEW**.

Submitting plots to a printer/plotter

You can submit a plot to any printer/plotter on your network, whether or not it is physically connected to your PC. MicroStation lets you use a variety of methods to submit plots or plotfiles to the preferred printer on your system. These include:

- Create a plotfile and copy this to the printer (at the system prompt).

- Send the plot directly to a locally connected printer through the parallel port without first creating a plotfile. You can do this either via the Plot dialog box or the key-in window.

- Via the Plot dialog box, send the plot directly to the system printer (either attached to your PC or elsewhere on the network).

➤ **To send a plotfile to a plotter through a parallel port**

1. Open a DOS window (session).

2. At the system command line, enter:

   copy /b <plot_file> <port>

   where:

   `plot_file` — is the plotfile to be sent.

   `port` — is the parallel port on your system to which the plotter is connected; for example, **lpt1: or lpt2:** (Wintel systems) or `prn` (DEC Alpha).

   `/b` — switch specifying that the file is binary.

➤ **To send a plot to a plotter through a parallel port without creating a plotfile**

1. From the File menu, choose Print/Plot.

   The Plot dialog box opens.

2. In the dialog box, adjust the controls as desired. (For more information, see the procedures under "Creating a plotfile from the Plot dialog box" on page 8-1.)

3. In the dialog box's icon bar, click the Plot icon.

   or

   From the dialog box's File menu, choose Plot.

   The Save Plot As dialog box opens.

4. In the Name field, key in the name of the parallel port, for example, **lpt1:** or **lpt2:** (Wintel systems) or `prn` (DEC Alpha).

5. Click OK.

➤ **To send a plot directly to a plotter (without creating a plotfile) from the Key-in window**

1. In the Key-in window, key in:

PLOT <configuration | VIEW view_number | FENCE> <output_device>

where:

configuration — identifies the plot configuration file upon which to base the plot.

view_number — is the view number where VIEW is specified.

Output_device — specifies the destination plotter. For example, **lpt2:** or **\\printman\laserjet1**. **Output_device** may be omitted when "printer.plt" is the plotter driver file.

To use the Windows system printer, MicroStation provides a special plotter driver file "printer.plt."

➤ **To send the plot directly to the system printer**
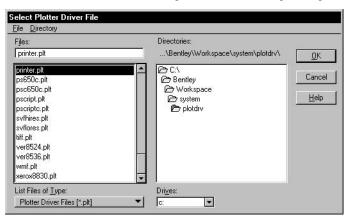
1. From the File menu, choose Print/Plot.
The Plot dialog box opens.

2. In the dialog box's icon bar, click the Plotter Driver icon.
or
From the dialog box's Setup menu, choose Driver.
The Select Plotter Driver File dialog box opens.

3. In the Files list box, select the plotter driver file, "printer.plt."

4. (Optional) Adjust other plotting settings as described in steps 4-5 in "Creating a plotfile from the Plot dialog box" on page 8-1.

5. In the Plot dialog box's icon bar, click the Plot icon.

or

From the Plot dialog box's File menu, choose Plot.

✍ In order to print in Windows, you must have a printer configured.

The exception is when MicroStation sends the plot data to a parallel port to which a plotter is attached.

Using Plotting key-ins

You can submit plots to a printer/plotter via key-ins from the key-in window. Plotting key-in commands provide additional key-in options for customizing your plot files to specific criteria. You can use these key-ins directly, in BASIC macros or in MDL applications.

Plotting key-in commands are provided by the "plotcfg.ma" plotting application.

One-step Commands

You can use one-step commands to set up and produce a plot in one step. These commands extend the existing plotting key-in:

PLOT <configuration> <filename>

used to bypass the Plot Preview dialog box. All plotting key-ins are accessible from the MDL application "plotcfg.ma." You can execute the one-step commands before or after loading "plotcfg.ma."

For example, you can create plotfiles of the contents of a view or fence as follows.

➤ **Using a key-in to create a plotfile from the contents of a fence**

1. Use the *Place Fence* tool to place a fence around the part of the design to be plotted.

2. In the key-in window, key in:
   PLOT FENCE [output_filename]

   where `output_filename` is the name of the plotfile.

➤ **Using a key-in to create a plotfile of the contents of a view**

1. Set up a view to display the part of the design to be plotted.

2. In the key-in window, key in:
   PLOT VIEW [view_number] [output_filename]

   where:

   `view_number` — is the number of the view.

   `output_filename` — is the name of the plotfile.

Where the active printer is the system printer, you can send the plot directly to the printer by omitting the `output_filename` parameter.

For a full list of one-step key-in commands, refer to "One-step commands" on page A-1.

Combined Commands

You can use Combined Commands to both set up and produce a plot. For example, you can use **SET PLOTTER**, **PLOT ENTITY FENCE** and **PLOT SCALE** to set up the plot, and then use **PLOT EXECUTE** to produce the plot. Most of the combined key-ins require that "plotcfg.ma" be loaded prior to the key-in being executed. Even those key-ins that are valid prior to "plotcfg.ma" being loaded result in it being loaded.

You can key in multiple combined commands, separating each with a semi-colon (;).

➤ **Using combined commands to select the postscript plotter, contents of a fence, scale and then to create the plotfile**

1. Use the *Place Fence* tool to place a fence around the area to be plotted.

2. In the key-in window, key in the following:

SET PLOTTER pscript.plt; PLOT ENTITY fence; PLOT SCALE 2:1; PLOT EXECUTE section1.eps

to create a postscript plotfile named "section1.eps" containing a 2:1 scale plot of the contents of a fence.

For a full list of combined commands key-ins, see "Combined Commands" on page A-2.
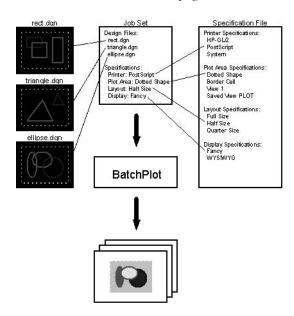
9 Batch Printing/Plotting

BatchPlot lets you create plots of sets of design files. It is not a replacement for the standard plot dialog box. It is simply a plotting utility that focuses on plotting multiple design files versus plotting the active design file.

Overview

BatchPlot is a utility for plotting and re-plotting related sets of design files in reusable ways. **Plot specifications** are the mechanism for describing these reusable ways. **Job sets** are used to identify, save, recall and print related sets of design files.

Each job set plots at a selected scale, so it is important to consider this when setting up your drawings in the first place. If all related drawings can be plotted at one scale, BatchPlot can create a full set of drawings in one step. A simple method for setting up your drawings to do this is described in "Using a 1:1 scale title block (recommended method)" on page 5-3.

Plot specifications

A plot specification is a named group of instructions describing how to perform a certain step in the plotting process. A given specification does not encompass the entire plotting process — just a particular step in it.

Plot Specification Types

BatchPlot supports four specification types corresponding to four plotting process steps:

Specification Type	Description
Printer	Describes the target printer, paper size and post-processing options. See "Printer Specifications" on page 9-13.
Plot Area	Selects the portion or portions of the design file to plot. A plot area specification essentially describes a view or the equivalent of one or more fences in a view. See "Plot Area specifications" on page 9-19.
Layout	Places a representation of the given plot area on the paper at the specified size and position. See "Layout specifications" on page 9-25.
Display	Identifies a pen table and/or control view attributes (for example, to turn text nodes and construction lines off) and plots options (displays fence boundary and plot border). See "Display specifications" on page 9-28.

Plotting Sets of Design Files

In order to plot a set of design files, the following steps are necessary to select the design files and specifications for plotting.

➤ **General procedure for plotting a Set of Design Files**

1. Select the design files to plot. See "Selecting design files" on page 9-3.

2. Select the specifications to control the plotting. See "Viewing and selecting specifications to control plotting" on page 9-4.

3. Plot the files. See "To print design files" on page 9-5.
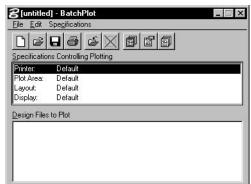
You can carry out all the above operations from the BatchPlot dialog box.

Selecting design files

Your first step in batch plotting is to select the design files that you wish to plot.

➤ **To select design files to plot**

1. From the File menu, choose Batch Print/Plot.

The BatchPlot dialog box opens.

BatchPlot dialog box.

2. In the BatchPlot dialog box, click the Add Design Files icon.

or

From the dialog box's Edit menu, choose Add Files.

The Select Design Files to Add dialog box opens.

3. Navigate to the desired directory and select the names of the files you want to plot.

You can select multiple files by holding down the <Ctrl> key as you select the files and then proceed to step 4.

Alternatively, you can double-click each file to send it to the Selected Files list box and then proceed to step 5.

4. Click Add

The filenames display in the Selected Files list box at the bottom.

5. Click Done.

The Select Files to Add dialog box closes and the selected files are added to the Design Files to Plot list box of the BatchPlot dialog box.

✍ You can also add files to the job set by dragging their names from Windows Explorer onto the BatchPlot dialog box.

Arranging the plotting order

When BatchPlot prints a job set, it prints the files in the order in which they appear in the list box. You can rearrange the order of the files in the list.

➤ **To rearrange the plotting order**

1. In the BatchPlot dialog box, highlight the file(s) that you wish to move. To select multiple files, hold down the <Ctrl> key when selecting the files.

2. From the BatchPlot dialog box's Edit menu, choose:

   Move Files to Top — to move selected files to the top of the list.

   Move Files Up — to move selected files up one position.

   Move Files Down — to move selected files down one position.

   Move Files to Bottom — to move selected files to the bottom of the list.

Viewing and selecting specifications to control plotting

Specifications are used to control the Printer, Plot Area, Layout or Display.

➤ **To view or change the properties of a specification**

1. In the Specifications Controlling Plotting list box, select the specification type (Printer, Plot Area, Layout or Display).

2. Click the Specification Properties icon.

   or

   From the Specifications menu, choose Properties.

   The appropriate properties dialog box opens.

3. (Optional) Make any required changes to the settings.

4. Click OK.

➤ **To select a specification to control plotting**

1. In the Specifications Controlling Plotting list box, select the specification type (Printer, Plot Area, Layout or Display).
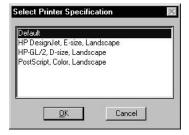
2. Click the Select Specification icon.

    or

    From the Specifications menu, choose Select.

    The Select Specification dialog box opens showing all existing specifications of the selected type.

Select Printer Specification dialog box

3. Select the desired specification.

4. Click OK.

    The Select Specification dialog box closes and the new specification replaces the selected one in the Specifications Controlling Plotting list box.

✎ BatchPlot automatically selects a specification of each plot specification type. You can select a different plot specification for each type.

Printing the selected design files

When you have selected the specifications for plotting, you can print the selected design files.

➤ **To print design files**
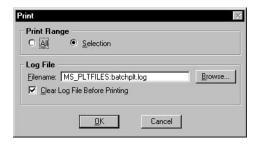
1. In the Design Files to Plot list box, select the files to be printed.

    If you are printing all the files in the list, skip to Step 2.

2. In the BatchPlot dialog box, click the Print icon.

or

From the BatchPlot dialog box's File menu, choose Print.

The Print dialog box opens.

Print dialog box

3. (Optional) To print all the files in the list, select the All button in the Print Range section.

4. (Optional) To setup a system printer, click the Set Up System Printer button. The Set Up System Printer button appears in the Print dialog box only if the active printer specification represents a native operating system printer.

The Print Setup dialog box opens.

5. Click OK to accept the printer setup changes and return to the Print dialog box.

6. (Optional) Turn off Clear Log File Before Printing.

By default, BatchPlot creates a log file describing the processing that occurs during actual printing, including any errors that occurred. You can disable the creation of the log file and redirect the log messages to the screen by clearing the Filename field. You can also instruct BatchPlot to append its logging messages to the end of an existing log file (by turning off Clear Log File Before Printing).

7. Click OK to start printing.

A progress window opens showing the name and sequence number of the currently printing file. You can stop the job by resetting.

Plotting Errors

When BatchPlot prints a set of design files, various specification-related errors may occur, including (but not limited to):

- The saved view, shape element or cell referenced by a plot area specification may not exist.

- The shape element or cell referenced by a plot area specification may be perpendicular to the reference view, so it appears as a line in the view.

- The plot size, calculated from the plot area and the scale or size, may exceed the size of the paper selected by the printer specification.

- The plot origin specified by a layout specification, coupled with the plot size calculated from the plot area and the plot scale, may shift part of the plot area off the page.

When BatchPlot detects an error, it stops processing the current plot boundary in the current design file, but continues processing with the next plot boundary or file.

Job Sets

A **job set** is an ordered list of design files to be plotted, along with references to the plot specifications that control how they are plotted. Job sets are stored in job set files (.job) with arbitrary names and locations. For example, a job set file named "myhouse.job" might represent the set of plans, elevations and detail sheets for a particular house. It lists all the master design files representing those plans, elevations and detail sheets and the names of the specifications describing how they should be plotted.

BatchPlot allows you to reassign the specifications assigned to a job set at any time, making it easy to plot variations of the set. Files can be added, deleted or moved up or down within the job set.
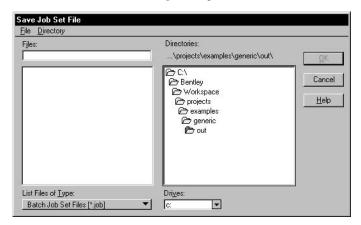
➤ **To create a new job set**

1. Use the BatchPlot dialog box to select the design files to be included and to select the plot specifications required.

2. In the BatchPlot dialog box, click the Save icon.

    or

    From the BatchPlot dialog box's File menu, choose Save.

    The Save Job Set File dialog box opens.

3. In the Files field, key in the name.

4. Click OK.
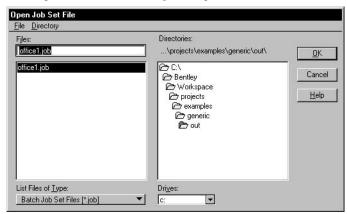
    The Save Job Set File dialog box closes.

➤ **To load an existing job set**

1. In the BatchPlot dialog box, click the Open icon.

   or

   From the BatchPlot dialog box's File menu, choose Open.

   The Open Job Set File dialog box opens.

2. Select the desired job set.

3. Click OK.

   The Open Job Set File dialog box closes and the selected job set's member design files are listed by name in the Design Files to Plot list box. As well, all plotting specifications associated with the job set also display in the Specifications Controlling Plotting list box.

✍ BatchPlot does not embed plot specifications in job set files. Instead, job sets reference (by name) specifications are stored in a separate specification file. This separation enables the easy sharing of specifications across multiple job sets. It also simplifies maintenance. New specifications are automatically available for use in all job sets. As well, corrections or additions to specifications are automatically propagated to sets that reference them.

Using Plot Specifications

Plot specifications control the following aspects of plotting:

- Printer (see "Printer Specifications" on page 9-13)

- Plot Area (see "Plot Area specifications" on page 9-19)

- Layout (see "Layout specifications" on page 9-25)

- Display (see "Display specifications" on page 9-28)

Using plot specifications you can set up a system for plotting the drawings for a particular project. Once set up, you do not need to enter the drawing file to create a plot of it. You can produce a plot of one or more drawings via BatchPlot.

Managing Plot Specifications

Using the BatchPlot Specification Manager dialog box, you can create, copy, rename and delete plot specifications.

✍ Whenever you create, rename, copy or edit a plot specification, BatchPlot automatically assigns it to the current job set.

Rename a plot specification

You can rename a plot specification but you should be careful when doing this. You should take into account that changing the name of a specification:

- May cause the descriptive name to not match the contents of the specification.

- Affect other job sets that have referenced the plot specification.

➤ **To rename a plot specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

Batch Plot Specification Manager dialog box

2. In the Types list box, select the type of specification to rename.

3. In the Specifications list box, select the specification to rename.

4. Click the Rename button.

   The appropriate Rename Specification dialog box opens.

5. In the Name field, key in the new name.

6. Click OK.

Create a new plot specification

For each specification Type, you can define multiple Specifications. For example, you may have a choice of printers/plotters that you can access. Also, you may have several page sizes that you use. You can create specifications to account for each combination that is required.

➤ **To create a new plot specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select the type of specification to create.

3. Click the New button.

   The appropriate New Specification Name dialog box opens.

4. In the Name field, key in the name for the new specification.

5. Click OK.

   The appropriate Properties dialog box opens so you can define the new specification. See "Defining Plot Specifications" on page 9-13.

Copying a plot specification

To create a new specification that is similar to that of one existing, you can copy the existing specification. Once copied, you can make any adjustments that you require.

➤ To copy a plot specification

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select the type of specification to copy.

3. In the Specifications list box, select the specification to copy.

4. Click the Copy button.

   The appropriate New Specification Name dialog box opens.

5. In the Name field, key in the new name.

6. Click OK.

   The appropriate Properties dialog box opens so you can modify the copied specification. See "Defining Plot Specifications" on page 9-13.

Deleting a plot specification

Where a plot specification is no longer wanted, you can delete it. Be careful when deleting plot specifications that other job sets have not referenced them. Once deleted, they will not be available for any other job sets that may have referenced them.

➤ **To delete a plot specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

    or

    From the BatchPlot dialog box's Specifications menu, choose Manage.

    The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select the type of specification to delete.

3. In the Specifications list box, select the specification to delete.

4. Click Delete.

Defining Plot Specifications

You can define the properties for each plot specification of each specification type (Printer, Plot Area, Layout, Display).

✍ When you modify specifications, keep in mind that they are shared across job sets. If you edit a specification, your changes affect all job sets that reference it. Similarly, if you delete or rename a specification used by another job set, BatchPlot automatically selects a new specification of the appropriate type the next time it opens that other job set.

Printer Specifications

A Printer specification selects a MicroStation plotter driver, a paper size, a paper orientation and specifies post-processing options.

Define printer specifications

You can create new printer specifications via the BatchPlot Specification Manager dialog box.

➤ **To create a new Printer specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

    or

    From the BatchPlot dialog box's Specifications menu, choose Manage.

    The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Printer.

3. Click the New button.

The New Printer Specification Name dialog box opens.

4. In the Name field, key in the name for the new specification.

5. Click OK.
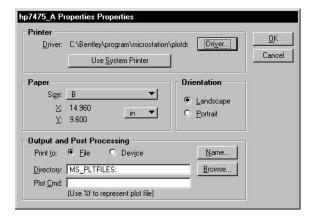
The New Printer Specification Name dialog box closes and the Properties dialog box opens for the new printer specification.

6. Click the Driver button.

The Select Plotter Driver File dialog box opens.

7. Select the required plotter driver file.

8. Click OK.

The Properties dialog box opens for the selected plotter driver file to be associated with the newly named specification.

Properties dialog box for the new printer specification.

➤ **Setting the properties in the Properties dialog box**

1. From the Size option menu, choose a paper size.

2. From the units option menu, choose the dimension readout in the desired units.

3. In the Orientation section, select the desired paper orientation (or plot rotation).

4. In the Output and Post Processing section's "Print to" options, select:

File — to create plotfiles.

Device — to print directly to a device.

See "Output and Post Processing options" on page 9-15.

5. Click OK to accept the modifications to the Printer Specification Properties.

Output and Post Processing options

In the Output and Post Processing section of the printer Properties dialog box, you can choose between creating plotfiles (optionally with a post processing system command) and printing directly to a device.

➤ **To create plotfiles**

1. In the Output and Post Processing section's "Print to" options, select File.

Directory and Plot Cmd fields appear. The default output directory is that specified by the configuration variable MS_PLTFILES.

2. (Optional) To change the default output directory, key in a new value.

or

Click the Browse button to open the Select Plot File Output Directory dialog box. Use this dialog box to select a new directory, then click OK.

3. (Optional) To specify an operating system command to be executed for each generated plot file, key in the command into the Plot Cmd field, using "%f" to represent the plot file name.

4. (Optional) To change the format of the filenames assigned to plotfiles, click the Name button and use the controls in the Plot Filename Format dialog box. When finished, click OK to return to the Printer Properties dialog box.

By default, plotfiles are named after the design file with a three-digit numeric extension. Multiple plotfiles from the same design file are distinguished by incrementing this numeric extension (for example, abc.000, abc.001, abc.002).

When you click the Name button in the Output and Post Processing section, three of the Format options allow you to specify a file extension in the Extension field.

Choosing Custom activates the String field for entry of a custom format. This format can be literal characters plus any combination of the following substitution strings.

Substitution String	Replaced By
%j	Base job set file name ("batchplt" if job set is untitled)
%d	Base design file name (no directory or extension)
%e	Contents of Extensions field
%p	Three-digit plot counter starting with 000. This counter is not reset between design files in a job set.

Substitution String	Replaced By
%b	Three-digit per-design-file plot boundary counter starting with 000. This counter resets between design files in a job set.
%x	Design file extension.

If you have design files with the same base name but different extensions, you can use the %x substitution string to differentiate the plot file names. This lets you include the file extension in the plotfile name.

Take, for example, the filenames `abc.dgn`, `abc.hln`, `abc.s01`. Here you could specify in the Plot Filename Format dialog box that the Extension be `ps` and that the String be `%d-%x.%e`. This would generate plot filenames `abc-dgn.ps`, `abc-hln.ps`, `abc-s01.ps`.

Your other choice, in the Output and Post Processing section of the Default Properties dialog box, is to print directly to a device.

➤ **To print directly to a device**

1. In the Output and Post Processing section, select Print to Device.

   The output directory and post-processing command fields clear and a Port field displays.

2. In the Port field, key in the port name (for example, **lpt1: or \\server\printer**).

   Keying a name in this field has the same effect as adding a default_outFile=filename record to a plotter driver (.plt) file.

Modifying a printer specification

You can modify an existing Printer specification from the BatchPlot Specification Manager dialog box. Also, you can modify the selected Printer specification from the BatchPlot dialog box.

✐ Remember, when you modify a Printer specification, it affects all job sets that reference that particular specification.

➤ **To modify an existing Printer specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

or

From the BatchPlot dialog box's Specifications menu, choose Manage.

The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Printer.

3. In the Specifications list box, select the specification that is to be modified, then click Properties.

or

In the Specifications list box, double-click the specification to be modified.

The selected printer specification's Properties dialog box opens.

4. Make changes as required to the settings.

5. Click OK to accept modifications to Printer Specification Properties.

The printer specification's Properties dialog box closes.

➤ **Modify the selected printer specification from the BatchPlot dialog box**

1. In the Specifications Controlling Printing list box in the BatchPlot dialog box, double-click Printer.

   or

   Select the Printer Specification entry in the list box and then from the Specifications menu, choose Properties.

   The printer Properties dialog box opens.

Printer Properties dialog box.

2. Make changes as required to the settings.

3. Click OK to accept modifications to Printer Specification Properties.

   The printer specification's Properties dialog box closes.

Plot Area specifications

A Plot Area specification describes how BatchPlot determines which portion of each design file to print. The two basic options are:

• A view — may be one of the eight numbered views or it may be a saved view(s).

• A fenced area in a view — because fences are not stored in design files and because BatchPlot is essentially non-interactive once it starts printing, BatchPlot dynamically creates fences from particular boundary-defining elements, such as shapes or a cell (similar to placing a fence by snapping to the vertices of a shape). The Plot Area Specification Properties dialog box is used to define the characteristics of the shape elements or the name of the cell.

➤ **To create a new Plot Area specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Plot Area.

3. Click the New button.

   The New Plot Area Specification Name dialog box opens.

4. In the Name field, key in the name for the new specification (for example, Layouts).

5. Click OK.

   The New Plot Area Specification Name dialog box closes and the Properties dialog box for the new specification opens. The specification's name appears in the title bar of the dialog box (for example, Layouts Properties).

6. From the View option menu, choose the number of the desired view or Saved View.

   Designating a reference view determines the default plot options (which can be overridden, See "Display specifications" on page 9-28), viewing angle and, in 3D files, front and back clipping planes.

7. If you chose Saved View, key in the name of the Saved View(s).

   When you choose Saved View, a field appears adjacent the View option menu.

8. From the Method option menu, choose from:

View to plot a view or saved view(s) in each file.

Shape to plot an area bounded by a particular shape. When you select Shape, toggles appear that let you define the attributes of the shape (that is, the Level, Color, Style and Weight attributes).

Cell to plot an area bounded by a cell. When you select Cell, a Name field appears, into which you key in the name of the cell. The cell name may be a literal name (for example, "border") or regular expression (for example, "bdr.*" to match any cell name starting with "bdr").

9. (Optional) Turn off the Master File and/or Reference Files settings to limit the search for the boundary-defining shape or cell.

By default, BatchPlot searches each master file and all of its reference files to find the boundary-defining shape or cell. You restrict the search to specific reference files by typing their logical names (for example, "border") or filenames (without drives or directories, for example, "border.dgn") into the multi-line Reference Files field. As with cell names, reference file names may be literals or regular expressions.
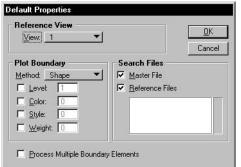
10. Turn on Process Multiple Boundary Elements to generate a plot for each boundary element found.

or

Turn off Process Multiple Boundary Elements to generate a plot for the first boundary element found only.

11. Click OK to accept the settings.

The Properties dialog box closes and the new specification becomes the selected Plot Area specification in the BatchPlot dialog box.

Appropriate fields appear in the Plot Boundary area when you choose Shape (left) or Cell (right)

Selecting multiple cells, reference files or saved views

When specifying the name of a cell, reference file or saved view, you can use regular expressions to choose multiple items. Regular expressions let you define strings with variables for matching the name(s) of the required items. They provide a string matching mechanism and are defined by the following rules:

Character	Meaning
c	any non-special character c matches itself.
\c	turn off special meaning of character c.
^	beginning of line.
$	end of line.
.	any single character.
:a	any alphabetic character [a-z|A-Z].
:d	any digit [0-9].
:n	any alphanumeric character [a-z|A-Z|0-9]. Also, a colon followed by a space matches any punctuation character.

Character	Meaning
[...]	any one of characters in ...; ranges like a - z are legal.
[^...]	any single character not in ...; ranges are legal.
s*	zero or more occurrences of string s (the preceding character).
s+	one or more occurrences of string s (the preceding character).
st	string s followed by string t.

Examples

Take, for example, the case where there are six reference files attached — *plot1.dgn*, *plot2.dgn*, *plot3.dgn*, *plota.dgn*, *plotb.dgn*, and *prelim.dgn*.

The following table shows those filenames that are matched when using various reference file name specifications.

Specify	To find the following
p.*	All six reference files. Finds all files beginning with the letter "p" followed by zero or more of any other characters.
plot:d.dgn	Three reference files — *plot1.dgn*, *plot2.dgn* and *plot3.dgn*. Finds all files beginning with "plot" followed by a single digit and then ".dgn."
plot:a.dgn	Two reference files — *plota.dgn* and *plotb.dgn*. Finds all reference files beginning with "plot" followed by a single alphabetic character and then ".dgn."
plot:n.dgn	Five reference files — all but *prelim.dgn*. Finds all reference files beginning with "plot" followed by a single alphanumeric character and then ".dgn."

Modifying a Plot Area specification

You can modify an existing Plot Area specification from the BatchPlot Specification Manager dialog box. Also, you can modify the selected Plot Area specification from the BatchPlot dialog box.

✍ Remember, when you modify a Plot Area specification, it affects all job sets that reference that particular specification.

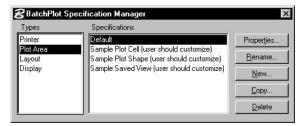➤ **To modify an existing Plot Area specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

    or

    From the BatchPlot dialog box's Specifications menu, choose Manage.

    The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Plot Area.

3. In the Specifications list box, select the specification that is to be modified, then click Properties.

    or

    In the Specifications list box, double-click the specification to be modified.

    The selected plot area specification's Properties dialog box opens.

4. Make changes as required to the existing settings.

5. Click OK to accept the modifications to the Plot Area Specification Properties.

    The Properties dialog box closes.

➤ **Modify a selected Plot Area specification from the BatchPlot dialog box**

1. In the Specifications Controlling Printing list box in the BatchPlot dialog box, double-click Plot Area.

    or

    Select the Plot Area specification entry in the list box and then from the Specifications menu, choose Properties.

    The appropriate Plot Area Properties dialog box opens.

2. Make changes as required to the existing settings.

3. Click OK to accept the modifications to the Plot Area Specification Properties.

The Properties dialog box closes.

Layout specifications

A Layout specification describes how BatchPlot determines the size and position of each plot on the selected sheet size.

BatchPlot supports five plot size options, which are selected via the Plot Size/Scale Method option menu in the Properties dialog box. Options are:

- **Maximize** —- makes each plot as large as possible given the paper size and orientation in the job set's Printer specification.

- **Scale** — specifies a scale factor that maps master units in the design file to physical units on the output media. The plot size is calculated from this scale and the size of the plot area.

- **% of Maximum Size** — specifies an integer between 10 and 100 inclusive to make each plot some percentage of its maximum possible size on the selected sheet size.

- **X Size** — specifies an explicit X size (width) for the plot. The plot's Y size is calculated from this X size and the plot area's aspect ratio.

- **Y Size** — specifies an explicit Y size (height) for the plot. The plot's X size is calculated from this Y size and the plot area's aspect ratio.

BatchPlot supports two Plot Origin options, which are selected via the Plot Origin Method option menu in the Properties dialog box. Options are:

- **Center** — centers each plot on the selected sheet.

- **Manual Offset**— specifies explicit X and Y offsets for the plot. These offsets are relative to the selected sheet's lower-left margin.

✎ The Center origin option and the Maximize and % of Maximum size options are safe options. They never cause print-time errors. Certain combinations of the other options, combined with a particular plot area's size or shape, may result in print-time errors.

Example of Print-time Errors

Consider a design file representing a map defined in terms of meters. A plot scale of 1 (meaning one master design unit maps to

one mm or inch) almost certainly results in a "plot size too large" error when used with that file. That same scale factor might work fine with a design file representing some small mechanical part. See "Plotting Errors" on page 9-7.

➤ **To create a new Layout specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

    or

    From the BatchPlot dialog box's Specifications menu, choose Manage.

    The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Layout.

3. Click the New button.

    The New Layout Specification Name dialog box opens.

4. In the Name field, key in the name for the new specification (for example, A4 portrait or 1"=20').

5. Click OK.

    The New Layout Specification Name dialog box closes and the Properties dialog box for the new specification opens.

The specification's name appears in the title bar of the dialog box (for example A4 portrait Properties or 1"=20' Properties).

6. From the Plot Size/Scale Method option menu, choose the required method (see above for descriptions).

7. From the Plot Origin Method option menu, choose the required method (see above for descriptions).

8. From the Units option menu, select the desired units for the plotter output.

   The scale of the plot is calculated as master units per plotter output units. This setting defines the plotter output units.

9. Click OK to accept the settings for the new Layout specification properties.

   The Layout Properties dialog box closes and the new specification becomes the selected Layout specification in the BatchPlot dialog box

Modifying a Layout specification

You can modify an existing layout specification from the BatchPlot Specification Manager dialog box. Also, you can modify the selected Layout specification from the BatchPlot dialog box.

✍ Remember, when you modify a Layout specification, it affects all job sets that reference that particular specification.

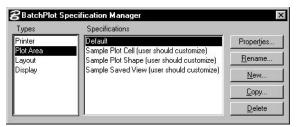➤ **To modify an existing Layout specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Layout.

3. In the Specifications list box, select the required specification, then click Properties.

   or

   In the Specifications list box, double-click the specification to be modified.

   The selected specification's Properties dialog box opens.

4. Make changes as required to the existing settings.

5. Click OK to accept the modifications to the Layout specification properties.

   The Properties dialog box closes.

➤ **To modify the selected Layout specification from the BatchPlot dialog box**

1. In the Specifications Controlling Printing list box in the BatchPlot dialog box, double-click Layout.

   or

   Select the Layout specification entry in the list box and then from the Specifications menu, choose Properties.

   The appropriate Layout Properties dialog box opens.

2. Make changes as required to the existing settings.

3. Click OK to accept the modifications to the Layout specification properties.

   The Properties dialog box closes.

Display specifications

A Display specification controls the appearance of printed elements. You can specify a pen table that will resymbolize the plot or you can control one or more display options — the plotting equivalent of the MicroStation view attributes.

Each option has three possible states:

- As-is — causes BatchPlot to use the setting saved in the design file.

- On — turns the setting on.

- Off — turns the setting off.

➤ **To create a new Display specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Display.

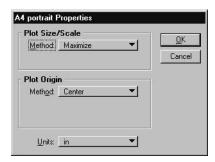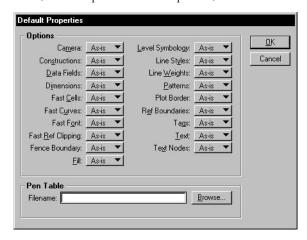
3. Click the New button.

   The New Layout Specification Name dialog box opens.

4. In the Name field, key in the name for the new specification (for example, No Fills).

5. Click OK.

   The New Display Specification Name dialog box closes and the Properties dialog box for the new specification opens.

The specification's name appears in the title bar of the dialog box (for example No Fills Properties).

Default Properties

Options

Camera:	As-is	Level Symbology:	As-is
Constructions:	As-is	Line Styles:	As-is
Data Fields:	As-is	Line Weights:	As-is
Dimensions:	As-is	Patterns:	As-is
Fast Cells:	As-is	Plot Border:	As-is
Fast Curves:	As-is	Ref Boundaries:	As-is
Fast Font:	As-is	Tags:	As-is
Fast Ref Clipping:	As-is	Text:	As-is
Fence Boundary:	As-is	Text Nodes:	As-is
Fill:	As-is		

OK
Cancel

Pen Table

Filename: [   ] Browse...

6. (Optional) Make the desired changes in the state of each option.

7. (Optional) To attach a pen table to resymbolize the plot, key in the name and location of a pen table in the Filename field or use the Browse button to select the pen table.

8. Click OK to accept the settings for the new Display specification properties.

   The Display Properties dialog box closes and the new specification becomes the selected Layout specification in the BatchPlot dialog box

Modifying a Display specification

You can modify an existing Display specification from the BatchPlot Specification Manager dialog box. Also, you can modify the selected Display specification from the BatchPlot dialog box.

✍ Remember, when you modify a Display specification, it affects all job sets that reference that particular specification.

➤ **To modify an existing Display specification**

1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

2. In the Types list box, select Display.

3. In the Specifications list box, select the required specification, then click Properties.

    or

    In the Specifications list box, double-click the specification to be modified.

    The selected specification's Properties dialog box opens.

4. Make changes as required to the existing settings.

5. Click OK to accept the modifications to the Display specification properties.

    The Properties dialog box closes.

➤ **To modify the selected Display specification from the BatchPlot dialog box**

1. In the Specifications Controlling Printing list box in the BatchPlot dialog box, double-click Display.

    or

    Select the Display specification entry in the list box and then, from the Specifications menu, choose Properties.

    The appropriate Display Properties dialog box opens.

2. Make changes as required to the existing settings.

3. Click OK to accept the modifications to the Display specification properties.

    The Properties dialog box closes.

Example of Three-state Toggle Options

To illustrate the benefits of these three-state options, consider a set of design files each having a different combination of on/off view attribute settings for text nodes, constructions and fill. Assume that you want to plot all of these files with text nodes and constructions hidden, but you want to plot filled areas as they appear in the files. (Fill may be on in some files and off in others.) To obtain the desired results, simply set the Text Nodes and Constructions options to Off and set the Fill option (and all other options) to As-is. As BatchPlot opens each design file, it turns off text nodes and constructions but it will leave all other settings alone.

10 Tutorial: Batch Plotting

In this tutorial, you will learn how to create specifications and a job set for plotting drawing files.

Overview

Setting up a project for batch plotting of a set of design files requires two main steps.

1. Set up the specifications for Printer, Plot Area, Layout and Display.

2. Create a job set that lists the design files to be plotted.

Once these steps have been completed, you can quickly create plots of the listed design files. You can select one or more from the list to plot or you can plot them all.

For the following exercises, you will be using sample design files "bplot1.dgn" and "bplot2.dgn," which are located in the "…\Workspace\projects\tutorials\intro\dgn" directory. Both are drawings of brackets inside a title block "A4Title.dgn," which is referenced to both drawings. In both design files, Drawing Composition was used to set up View 8 as a sheet view.

In each, the model is at half-scale with the title block at full-scale.

BatchPlot sample drawing files.
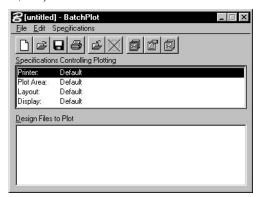
Preparation

You can run BatchPlot from any design file. For the following
exercise, we will open one of the sample files and set up the
relevant specifications.

➤ **Prepare for the exercise**

1. Open the design file bplot1.dgn.

2. From the File menu, choose Batch Print/Plot.

The BatchPlot dialog box opens. Currently it is untitled. That
is, no job set has been loaded.

BatchPlot dialog box

Define batch plotting specifications

Currently, all specifications are set as Default. Our first task is to create specifications that suit our needs. To do this, we will use the BatchPlot Specification Manager dialog box.

Create a Printer specification

We use the Printer specification to define the type of printer being used, the page size and the orientation. We also define whether the output is to a file on disk or directly to the plotting device.

➤ **Create the Printer specification**
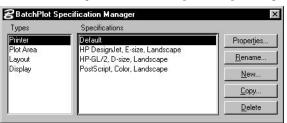
1. In the BatchPlot dialog box, click the Manage Specifications icon.

   or

   From the BatchPlot dialog box's Specifications menu, choose Manage.

   The BatchPlot Specification Manager dialog box opens.

*Batch Plot Specification
Manager dialog box*

2. If necessary, in the Types list box, select Printer.

3. Click the New button.

   The New Printer Specification Name dialog box opens.

4. In the Name field, key in **Postscript, B, Color, Landscape**.

   We are preparing a printer specification to plot on B size sheets in landscape orientation and in color. The A4 size title block will easily fit in the B size sheet.

5. Click OK.

The New Printer Specification Name dialog box closes and the Properties dialog box opens for the new printer specification.

6. Click the Driver button.

The Select Plotter Driver File dialog box opens.

7. Select the plotter driver file "psc650c.plt."

8. Click OK.

The Properties dialog box opens for the selected plotter driver file to be associated with the newly named specification.

Properties dialog box for the new printer specification.

We will now set the printer properties in the Properties dialog box.

➤ **Set the printer properties**

1. From the Paper option menu, choose B.

2. If necessary, in the Orientation section, turn on Landscape.

3. In the Output and Post Processing section, Turn on File.

We will print to File.

4. Click OK.

The dialog box closes. Notice that the Printer specification just completed is now the current setting in the BatchPlot dialog box.
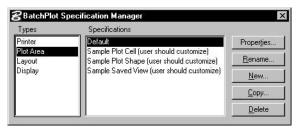
With the printer specification completed, you will now specify the area to be plotted in each design file.

Create a Plot Area specification

We use the Plot Area specification to define the View and Method that is used to select the area of the design file to be plotted.

➤ **Create the Plot Area specification**

1. In the BatchPlot Specification Manager dialog box's Types list box, select Plot Area.

2. Click the New button.

   The New Plot Area Specification Name dialog box opens.

3. In the Name field, key in **A4 title block**.

4. Click OK.

   The New Plot Area Specification Name dialog box closes and the Properties dialog box for the new specification opens.

5. From the View option menu, choose 8.

   In both drawing files, the drawing is set up in View 8.

6. From the Method option menu, choose Shape.

   Attribute toggles appear below the option menu to let you set the attributes for the shape that will define the plotting area.

   We will use the chain-dotted rectangle of the title block (in reference file A4Title.dgn) to define the plotting area for each file.

7. Turn on Level and in its field, key in **1**.

   The shape is on level 1 of the file A4Title.dgn.

8. Turn on Style and in its field, key in **7**.

   This limits the search for the shape to that created from the Type 7 line style.

9. Turn off Master File.

    We will limit the search for the plot area shape to the reference file only.

10. Turn on Reference Files.

11. In the Reference Files list box, key in **A4Title.dgn**.

    This limits the search for the shape to that file only.

12. Click OK.

    The dialog box closes. Notice that the defined Plot Area specification is now the current setting in the BatchPlot dialog box.

Plot Area Properties dialog box

Create the Layout specification

With the plot area defined, we now specify the layout on the selected sheet including the scale of the plot. In this case, the plots will be created full-size as the title blocks are drawn at full size. The drawing elements have been referenced in to the title block at half scale and will plot correctly at that scale if we plot the title blocks at their full size.

➤ **Create the Layout specification**

1. In the BatchPlot Specification Manager dialog box's Types list box, select Layout.

2. Click the New button.

    The New Layout Specification Name dialog box opens.
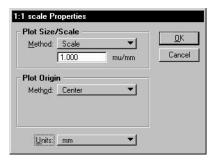
3. In the Name field, key in **1:1 scale**.

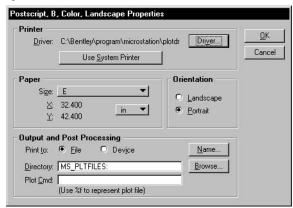4. Click OK.

    The New Layout Specification Name dialog box closes and the Properties dialog box for the new specification opens.

5. From the Units option menu at the bottom of the dialog box, select mm.

    This setting sets the plotter's units. Master Units for our design are mm, so selecting mm as the plotter's units will make it easier to set the scale.

6. From the Plot Size/Scale Method option menu, choose Scale.

    A scale field opens below the option menu.

7. In the Scale field, key in **1**.

8. If necessary, from the Plot Origin Method option menu, choose Center.

9. Click OK to accept the settings for the new Layout specification properties.
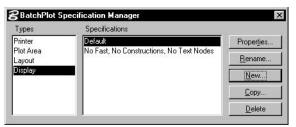
The Layout Properties dialog box closes and the new specification becomes the selected Layout specification in the BatchPlot dialog box.

Define the Display specification

Finally, before selecting the design files to plot, we will set the Display properties specification. This is similar to the View Attributes except that it affects the plot output. For our exercise, we will turn off the Plot Border.

➤ Create the Display specification

1. In the BatchPlot Specification Manager dialog box's Types list box, select Display.

2. Click the New button.
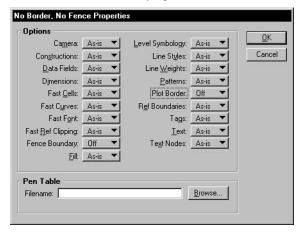
The New Layout Specification Name dialog box opens.

3. In the Name field, key in **No Border, No Fence**.

4. Click OK.

   The New Display Specification Name dialog box closes and the Properties dialog box for the new specification opens.

5. From the Plot Border option menu, choose Off.

6. From the Fence Boundary option menu, choose Off.

No Border, No Fence Properties

Options

Camera:	As-is	Level Symbology:	As-is	OK
Constructions:	As-is	Line Styles:	As-is	Cancel
Data Fields:	As-is	Line Weights:	As-is	
Dimensions:	As-is	Patterns:	As-is	
Fast Cells:	As-is	Plot Border:	Off	
Fast Curves:	As-is	Ref Boundaries:	As-is	
Fast Font:	As-is	Tags:	As-is	
Fast Ref Clipping:	As-is	Text:	As-is	
Fence Boundary:	Off	Text Nodes:	As-is	
Fill:	As-is			

Pen Table
Filename: [   ] Browse...

7. Click OK.

   The Display Properties dialog box closes and the new specification becomes the selected Layout specification in the BatchPlot dialog box.
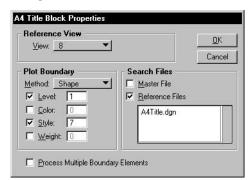
8. Close the BatchPlot Specification Manager dialog box.

Choose the design files and save the Job Set

With the setting up of the plotting specifications completed, we will now select the design files to be plotted and create the Job Set.

➤ **Select the design files for plotting**

1. In the BatchPlot dialog box, click the Add Design Files icon.
   The Select Design Files to Add dialog box opens.

2. Navigate to the "…\Workspace\projects\tutorials\intro\dgn" directory.

3. While holding down the <Ctrl> key, select "bplot1.dgn" and "bplot2.dgn."
   Both files are highlighted.

4. Click the Add button.

Both files appear in the Selected Files list box.

5. Click Done.
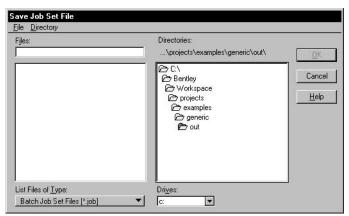
The Select Design Files to Add dialog box closes. The selected design files now appear in the Design Files to Plot list box in the BatchPlot dialog box.

With everything set up, ready to plot, we should first save the settings as a Job Set.

➤ **Save the Job Set**

1. In the BatchPlot dialog box, click the Save icon.

The Save Job Set File dialog box opens.

2. In the Files field, key in **brackets**.

By default, the extension ".job" is added to the file name.

3. Click OK.

The Save Job Set File dialog box closes.
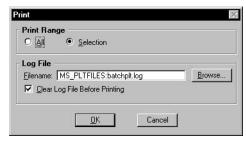
You can now plot the selected design files. We have specified that the output is to File, so plot files will be created on disk. Because we did not specify a particular directory, they will be stored in the directory pointed to by the configuration variable MS_PLTFILES.

➤ **Plot the selected design files**

1. In the BatchPlot dialog box, click the Print icon.

The Print dialog box opens.

Print dialog box

➤ **Print**

Print Range

○ All   ● Selection

Log File

Filename: MS_PLTFILES:batchplt.log   Browse...

☑ Clear Log File Before Printing

OK   Cancel

---

2. If necessary, select the All button in the Print Range section.

3. Click OK to start printing.

A progress window opens, showing the name and sequence number of the currently printing file. You can stop the job by resetting.

In the status bar (right side) the message "BatchPlot: files:2, plots:2, failures:0" should appear at the completion of plotting.

Re-using Job Sets

In the future, when you require a plot of these design files, you can simply open the job set file "brackets.job" from the BatchPlot dialog box. The following exercise shows this. First, clear the existing settings.

➤ **Clear the existing Job Set**

1. From the BatchPlot dialog box's File menu, choose New.

All details of the job set are cleared.

You can recall an existing Job Set at any time. It is not necessary to have open any of the design files required to be plotted. Each file is opened prior to plotting. At the completion of plotting, you are returned to the file in which you were working prior to plotting.

➤ **Open the Job Set created previously**

1. In the BatchPlot dialog box, click the Open icon.

The Open Job Set File dialog box opens.

2. In the Files list box, select brackets.job.

3. Click OK.

The dialog box closes and the Job Set settings are loaded in the BatchPlot dialog box.

Using this method lets you quickly replot groups of files or select one or more from a group without needing to enter each file and recreate the plotting parameters.

11 Plot Resymbolization and PenTables

This chapter introduces the general procedures for resymbolizing your design in the printed or plotted output.

Introduction

MicroStation's Printing and Plotting Pen Tables let you generate different outputs from a single design file or combinations of design files. Output information, either in the form of a plot or as a plot file, can assume characteristics different from the original design file. The appearance of the design can be altered selectively at the level of individual element characteristics. This even includes selective exclusion of elements. Also, the pen table lets you specify the plotting order of groups of elements, elements of specific color or line weight and elements of a specific type. The plotting order of individual levels also can be controlled. In addition, the text string substitution feature lets you substitute specific strings of text with either a replacement text string or a string variable such as the date or time.

The **pen table** is a data structure that:

- Selectively alters the plotted appearance of elements.

- Determines the plotting order of elements.

- Specifies text string substitutions.

Collectively, these processes are referred to as **plot resymbolization**.

A pen table is stored in a **pen table file**. The pen table consists of a series of **sections** that are tested against each element in the design. When a match is found, the output options, such as line weight and color are applied to the element. The modified element then is converted into plot data, which in turn is written to the plotfile. At no time are the elements of the design or its reference files modified.

Plot resymbolization

Resymbolization is the process of changing characteristics of elements within a design file. When these changes are applied to plotted output, the process is referred to as plot resymbolization. MicroStation uses pen tables to control plot resymbolization. Pen tables let you re-map any of the characteristics associated with design file elements including text for the plotted output. These changes are made outside the design file but can be applied to the design file. That is, once specified, you can retain them for future use. You can store multiple pen tables that can be applied to one or more design files. Using a pen table, you can specify many complex plot resymbolization steps. Similarly you can specify a single simple resymbolization step.

Pen tables

Pen tables are ASCII text files that contain instructions for resymbolizing the plotted output of design files. Instructions for resymbolization are contained in sections within the pen table. Pen table files are created automatically when you use the Pen Table dialog boxes to create or modify pen tables.

For each pen table section, there are element evaluation criteria and a set of output actions. During plot creation the pen table tests for the presence of specific types of elements and related element characteristics. If such elements are detected, the pen table will modify, enhance or eliminate these elements or their characteristics depending on the intended output. Criteria tested for include:

- Element type
- Files
- Weight
- Level
- Color
- Fill Color
- Style
- Class
- MSLink number (in external database)
- Entity number (in external database)
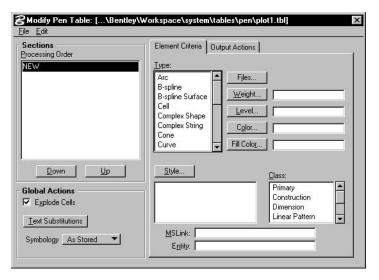
Evaluation of elements is based on the criteria that you specify. Any combination of these parameters can be evaluated. Many (such as level) can have multiple values and ranges of values.

➤ **To create a Pen Table**

1. From the Plot dialog box's PenTable menu, choose New.

    The Create Pen Table File dialog box opens.

    Pen tables are stored in the directory defined by the MS_PENTABLE configuration variable (typically MicroStation's "Workspaces/system/tables/pen/" directory).

2. In the Files field, key in the pen table file name.

    Only the file name need be entered. By default, the extension ".tbl" is added.

3. Click OK.

    The Modify Pen Table dialog box opens. The new pen table is loaded and pen table processing is activated.

When you create a pen table, MicroStation automatically inserts a single section called NEW. This section provides the minimum structure required in the pen table. You can either rename this section or insert a new one and delete NEW.

Modify Pen Table dialog box.

➤ **To load a previously existing pen table**

1. From the Plot dialog box's PenTable menu, choose Load.

    The Select Pen Table File dialog box opens.

2. In the Files list box, select the pen table.

3. Click OK.

   Focus returns to the Plot dialog box. The pen table is loaded and pen table processing is activated.

4. (Optional) To modify (or review the contents of) the loaded pen table, choose Modify from the Pen Table menu.

   The Modify Pen Table dialog box opens.

Modify Pen Table dialog box

All operations concerning pen tables can be carried out from the Modify Pen Table dialog box. Sections present in the currently loaded pen table are listed in the Processing Order list box. Its File and Edit menus let you create and edit both new and existing pen tables.

Using settings in the Element Criteria tab of the Modify Pen Table dialog box you can define the parameters for selecting elements in the design file (see "Element Criteria" on page 11-8). Having set the selection criteria, you can use the settings in the Output Actions tab to define the changes required for the plotted output (see "Output actions" on page 11-18).

You can define an individual parameter for element selection and based on that evaluation, specify modification of the identified element in the output plot file. Similarly, you can evaluate multiple parameters. However, when you specify more than one parameter, they are considered as a boolean AND operation. That is, elements for which you specify multiple parameter values must meet all specified criteria before they will be identified.

For example, if you specify Ellipse as the element type, you can specify one or more types of modifications to be applied to all ellipses in the design file. However, if you specify Ellipse, Level 40, Line weight 2, then only ellipses on level 40 with a line weight of 2 will be affected by the Output Action. All other ellipses in the design will be ignored by this section.

Subsequently, you can define changes that you want to apply to other types of Ellipses on other levels of the design file. Those changes will be applied without affecting the first section of changes. Note that elements modified by the present or previous sections cannot be changed in future sections.

➤ **To open the Modify Pen Table dialog box**

1. From the Pen Table menu, choose Modify.

   If a pen table is loaded, the Modify Pen Table dialog box opens.

   If no pen table is loaded, the Select Pen Table File dialog box opens. Proceed to the next step.

2. In the Files list box, select the pen table.

3. Click OK.

   The selected pen table is loaded and the Modify Pen Table dialog box opens.

File menu

Items in the Modify Pen Table dialog box's File menu are used to create new pen tables or to open existing pen tables for modification.

File menu item	Description
New	Opens the Create Pen Table file dialog box to create a new pen table.
Open	Opens the Select Pen Table File dialog box to open an existing pen table.
Save	Saves the current pen table file to disk.
Save As	Opens the Create Pen Table file dialog box to save the currently loaded pen table under a different name.
Exit/Unload	Closes the Modify Pen Table dialog box and unloads the pen table.

Edit Menu

Items in the Modify Pen Table dialog box's Edit menu are used to create and modify Sections in the currently loaded pen table.

Edit menu item	Description
Insert New Section Above	Inserts a new section above the currently selected section in the Processing Order list box.
Insert New Section Below	Inserts a new section below the currently selected section in the Processing Order list box.
Rename Section	Lets you rename the currently selected section.
Delete Section	Lets you delete the currently selected section.

Edit menu item	Description
Clear Types	Clears all element types currently selected.
Clear Classes	Clears all element classes currently selected.
Set All Types	Selects all element types.
Set All Classes	Selects all element classes.

Sections

Each section contains the definition for an individual resymbolization task or a group of related resymbolization tasks. These tasks can be applied to either a single element or a group of elements. In the Modify Pen Table dialog box, the Sections area displays the Processing Order of the sections that you have defined for the pen table.

Sections let you define resymbolization tasks separately from each other. You can specify virtually an unlimited number of resymbolization tasks of varying degrees of complexity. These resymbolization tasks can apply to any element that, while being similar to others, has at least one characteristic differentiating it from the others.

When creating sections, Element Criteria parameters define the elements to be processed and the Output Actions parameters define how the selected elements are treated when plotted.

➤ **To add a section**

1. In the Modify Pen Table dialog box's Processing Order list box, select an existing section at the point where you want to insert a new section.

2. From the dialog box's Edit menu, choose Insert New Section Above to insert the new section above the selected one.

   or

   From the dialog box's Edit menu, choose Insert New Section Below to insert the new section below the selected one.

   The Insert Section dialog box opens.

3. In the Name field, key in the new section's name.

4. Click OK.

➤ **To rename a section**

1. In the Modify Pen Table dialog box's Processing Order list box, select the section to rename.

2. From the dialog box's Edit menu, choose Rename Section.

   The Rename Section dialog box opens.

3. In the Name field, key in the new name.

4. Click OK.

➤ **To delete a section**

1. In the Modify Pen Table dialog box's Processing Order list box, select the section name to delete.

2. From the dialog box's Edit menu, choose Delete Section.

➤ **To save a modified pen table**

1. From the Modify Pen Table dialog box's File menu, choose Save.

   or

   To save the modified pen table in a different pen table file, choose Save As and then use the controls in the Save Pen Table As dialog box to specify the destination filename, directory and disk.

➤ **To modify a pen table section**

1. In the Modify Pen Table dialog box's Processing Order list box, select the section.

2. Set the Element Criteria for the selected section. Element Criteria specify which elements are selected for processing. See "To set the element criteria for a pen table section" on page 11-9.

   The list boxes used to specify element type, class and file criteria allow multiple, disjointed selections. <Ctrl>-click to

select multiple non-contiguous items. <Shift>-click to select a range of items (multiple contiguous items).

The dialog box's Edit menu has items for selecting and deselecting list box entries:

Edit menu item	Description
Clear Types	Clears selection of element types.
Clear Classes	Clears all element classes currently selected.
Set All Types	Selects all element types.
Set All Classes	Selects all element classes.

The fields used to specify element line weight, level and color criteria allow multiple, disjoint value ranges specified using commas and hyphens — for example, **2, 7, 11-14, 16-18, 21**.

3. Set the Output Actions for the selected section. Output Actions specify what is to be done once an element meets the section's criteria. See "To set the output actions for a pen table section" on page 11-18.

Processing Order

As you create sections, they are added to the Processing Order list box. During processing, each element is checked against the first section, then the second section and so on, through to the last section or until a match occurs. Once an element has been modified by a pen table section, it cannot be processed by another section. Therefore, the order of processing is important. This list determines the order of processing of sections. When necessary, you can rearrange the order of sections in the Processing Order list.

➤ **To rearrange the processing order of sections**
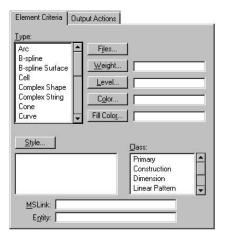
1. In the Processing Order list, identify the section to be moved.

2. Click the Down or Up button (below the list box) to move the selected section down or up one position in the list.

3. Repeat step 2 to further rearrange the order of the sections.

Element Criteria

Settings contained in the Element Criteria folder determine which elements in the design file are selected for processing. You do not have to define all settings. Where a setting is not defined, then all of that setting are included. For example, if you do not set a color or range of colors, then elements of all colors are selected.

➤ **To set the element criteria for a pen table section**

1. If the Element Criteria controls are not already displayed in the Modify Pen Table dialog box, click the Element Criteria folder tab.

Element Criteria controls

2. In the Type list box, select the element type(s).

3. In the Class list box, select the class(es).

4. Click the Files button to Identify the files that you want processed (see "Files" on page 11-10).

5. In the Weight field, key in the line weight value(s).

   or

   Click the Weight button and use the controls in the Select Weights dialog box to set the line weight value(s) (see "Weight" on page 11-11).

6. In the Level field, key in the level number(s).

   or

   Click the Level button and use the controls in the Select Levels dialog box to set the level number(s) (see "Level" on page 11-12).

7. In the Color field, key in the color number(s).

   or

   Click the Color button and use the controls in the Select Colors dialog box to set the color(s) (see "Color" on page 11-13).

8. In the Fill Color field, key in the fill color number(s).

    or

    Click the Fill Color button and use the controls in the Select Colors dialog box to set the fill colors (see "Fill Color" on page 11-14).

9. Click the Style button and use the controls in the Select Line Style dialog box to select the line style(s) (see "Style" on page 11-14).

10. In the MSLink field, key in the `mslink` column value(s) (see "Mslink column" in Chapter 5 in the *Administrator's Guide*).

11. In the Entity field, key in the entity number(s).

Files

This setting defines the design files that will be selected by the pen table for processing. You can select the active file and any reference files by their Slot Number. Reference files also can be selected by their Logical Names.

Using logical names for reference files produces more reusable pen tables. For example, you may have a number of design drawing files that use the same reference files, including the border file, but in different slot numbers. Selecting by logical names ensures that the correct resymbolization occurs for the reference files, irrespective of their slot numbers.

To identify the master file, you need to use its slot number, which is always zero because it does not have a logical name.

➤ **To select files for processing**

1. In the Element Criteria folder, click the Files button.

    The Identify Files dialog box opens.

2. In the Slot Numbers group box, click the Select button.

    The Select Files dialog box opens.

3. In the Select Files dialog box, select the required files.

4. Click OK.

    The dialog box closes and the selected files' slot numbers appear in the Slot Numbers field.

5. In the Reference Files Logical Names group box, click the Select button.

    The Select Files dialog box opens.

6. Select the required reference files.

7. Click OK.

The Select Files dialog box closes and the files are added to the Reference Files Logical Names list.

Where you know the reference file logical names, you can key in the logical name in the field below the list box. When necessary, you can replace existing entries or remove them.

➤ **Using a key-in to select a reference file**

1. In the Element Criteria folder, click the Files button.
The Identify Files dialog box opens.

2. In the field below the Reference File Logical Names list box, key in the logical name of the required reference file.

3. Click Add.
The file is added to the list.

➤ **Replacing a reference file in the list**

1. In the Identify Files dialog box, select the reference file logical name that is to be replaced.

2. Key in the new reference file logical name.

3. Click Replace.

➤ **Removing a reference file logical name from the list**

1. In the Identify Files dialog box, select the reference file logical name to be removed.

2. Click Remove.
The reference file logical name is deleted from the list.

Weight

This setting defines the line weights that will be selected by the pen table for processing. When defining the weight parameter for element evaluation, you can key in the values or you can use the Weight button to select the values graphically.

➤ **To select Weight values graphically**

1. In the Modify Pen Table dialog box's Element Criteria folder, click the Weight button.

2. The Select Weights dialog box opens.

3. Select the required weights by clicking on them. Clicking on a selection a second time deselects it.

   or

   Use the All or None buttons to select all weight values or no weight values, respectively.

4. Click OK.

   The selected values are entered in the Weights field.

Select Weights dialog box

Level

This setting defines the design file levels that will be selected by the pen table for processing. When defining the level parameter for element evaluation, you can key in the values or you can use the Level button to select the values graphically.

➤ **To select Level values graphically**

1. In the Modify Pen Table dialog box's Element Criteria folder, click the Level button.

2. The Select Levels dialog box opens.

3. Select the required levels by clicking on them. Clicking on a selection a second time deselects it.

   or

   Use the All or None buttons to select all level values or no level values, respectively.

4. Click OK.

The selected values are entered in the Level field.

Select Levels dialog box

Color

This setting defines the element colors that will be selected by the pen table for processing. When defining the color parameter for element evaluation, you can key in the values or you can use the Color button to select the values graphically.
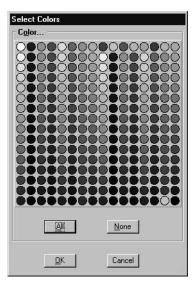
➤ **To select Color values graphically**

1. In the Modify Pen Table dialog box's Element Criteria folder, click the Color button.

2. The Select Colors dialog box opens.

3. Select the required colors by clicking on them. Clicking on a selection a second time deselects it.

or

Use the All or None buttons to select all color values or no color values, respectively.

4. Click OK.

The selected values are entered in the Color field.

Select Colors dialog box

Fill Color

This setting defines the fill colors that will be selected by the pen table for processing. When defining the fill color parameter for element evaluation, you can key in the values or you can use the Fill Color button to select the values graphically.

➤ **To select Fill Color values graphically**

1. In the Modify Pen Table dialog box's Element Criteria folder, click the Fill Color button.

2. The Select Colors dialog box opens.
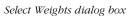
3. Select the required colors by clicking on them. Clicking on a selection a second time deselects it.

or

Use the All or None buttons to select all color values or no color values, respectively.

4. Click OK.

The selected values are entered in the Fill Color field.

Style

This setting defines the line styles that will be selected by the pen table for processing.

➤ **To select Style values**

1. In the Modify Pen Table dialog box's Element Criteria folder, click the Style button.

2. The Select Line Style dialog box opens.

3. Select the required line styles by clicking on them. Clicking on a selection a second time deselects it.

4. Click OK.

   The selected values are entered in the Style list box.

Global Actions

Global Actions are completed without regard to section definitions. They are applied across the entire design file and across all levels within the file. Global actions cannot be focused on any specific section. They provide three features for modifying your design's plotted output:

- Explode Cells
- Text Substitutions
- Symbology

Explode Cells

Explode Cells must be turned on if you want element criteria to be evaluated against individual elements making up the design file's cells. This is done without regard to the design levels containing the cells or the processing section. If Explode Cells is turned off, only the cell header is considered during element criteria evaluation.

Text Substitutions

You can use the text substitution feature to replace text in your design file for the plotted output. This can be with standard text string variables for text items such as the date, time or filename. Proposed replacement strings for the date, time, filename (full and abbreviated) are special strings that are converted to the required information during processing for plotting. When specifying these text substitutions, the system provides a suggested actual string. You can place an identical text string in the design file at the position that you want the text substitution. Alternatively, you can amend the actual string to match an existing text string in the design.

In addition, you can replace original "generic" text (text that is not standard design file information, such as date, time and file name) with replacement text strings.
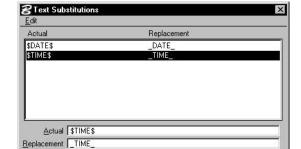
➤ **To define a text substitution entry for the filename, date or time**

1. In the design, place at least one string of text for which you want to substitute the filename, date or time for plotted output.

2. In the Modify Pen Table dialog box's Global Actions section, click the Text Substitutions button.

   The Text Substitutions dialog box opens.

3. From the Text Substitutions dialog box's Edit menu, choose from —Insert Abbreviated Filename, Insert Filename, Insert Date or Insert Time.

   Choosing one of these menu items inserts a list box entry. A special string appears in the Replacement field and a suggested string appears in the Actual field, as follows:

Edit menu item	Actual string	Proposed replacement string	Effect of leaving the proposed replacement string as is
Insert Abbreviated Filename	$FILEABBREV$	_FILEA_	Replaces the actual string with the filename of the active design file. The replacement filename is truncated to the size of the actual string.
Insert Filename	$FILE$	_FILE_	Replaces the actual string with the filename of the active design file. No truncation takes place.
Insert Date	$DATE$	_DATE_	Replaces the actual string with the current date, for example, "22 MAY 95."
Insert Time	$TIME$	_TIME_	Replaces the actual string with the current time, for example, "22:18:21."

4. (Optional) In the Actual field, edit the string to match an existing string in the design file.

5. Press <Return> to update the list box entry with the specified strings.

Text Substitutions dialog box with entries for time and date substitutions.

➤ **To insert a generic text substitution entry**

1. In the design, place at least one string of text for which you want to substitute other text for plotted output.

2. From the Text Substitutions dialog box's Edit menu, choose Insert New.

An entry labeled "Original" appears in the Actual field of the list box.

3. In the Actual field, replace "Original" with the string in the design that you want replaced for plotting purposes.

4. In the Replacement field, type the replacement string.

5. Press <Return> to update the list box entry with the specified strings.

✎ The entry automatically updates in the pen table when you close this dialog box, edit another string or insert a new string.

✎ If a replacement string or a portion of a replacement string has the format of a MicroStation configuration macro — that is, $(<text>) — the string undergoes substitution automatically to a preset value. This automatic substitution is called **expansion**. For example, the string User: $(_USTN_USERNAME) might expand to "User: JohnSmith." See "Configuration Variable File Syntax" in Chapter 4 in the *Administrator's Guide*.

If a string in configuration macro format is expanded to another configuration macro, substitution is iterative. That is, it continues until the resulting string does not contain a configuration macro.

Symbology

Symbology offers one of two selections against which to process your selected resymbolization criteria. Where element symbology has been enabled, you can choose to process elements by either of two methods:
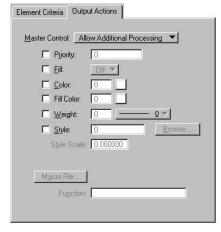
- As Stored — processes your criteria against the assigned element symbology in your design file. That is, elements are processed against their symbology, whether or not element symbology is enabled.

- As Displayed — processes the criteria against the design file as it is currently displayed. If element symbology is enabled, changing the displayed symbology of elements, then this "new" symbology is considered during processing.

Output actions

Output actions are associated with each pen table section. Output actions specify what is to be done once an element meets the section's Element Criteria.

➤ **To set the output actions for a pen table section**

1. If the Output Actions controls are not already accessible in the Modify Pen Table dialog box, click the Output Actions folder tab.

Output Actions controls

2. From the Master Control option menu, choose the required option (see "Master Control" on page 11-20).

3. (Optional) To set the order in which the element is plotted in relation to other elements, turn on Priority and key in the

desired priority value. The valid range is -2147483648 to 2147483647 (see "Priority" on page 11-20).

4. (Optional) To set the color in which the element is plotted, turn on Color, then key in the desired color value in the Color field.

    or

    Choose the desired color from the adjacent pop-up palette (see "Color" on page 11-21).

5. (Optional) To set the line weight with which the element is plotted, turn on Weight, then key in the desired color value in the Weight field.

    or

    Choose the desired line weight from the adjacent option menu (see "Weight" on page 11-21).

6. (Optional) To set the line style with which the element is plotted, turn on Style, then key in the name of the desired line style in the Style field.

    or

    Click the adjacent Browse button and select the desired line style in the Select Line Style dialog box (see "Style" on page 11-21).

7. (Optional) If the element is to be plotted with a custom line style (designated in step 6), key in the desired line style scale factor in the Style Scale field.

Master Control

Items in the Master Control option menu let you choose whether the element is modified, plotted or passed on to a custom BASIC macro, as follows:

Master Control option	Effect
Allow Additional Processing (default)	After the other Output Action controls have been applied to the selected element, any loaded MDL applications that desire to process the element further are afforded the opportunity.
No Additional Processing	No further processing is performed on the selected element after the other Output Action controls have been satisfied. No MDL applications that desire to process the element are afforded the opportunity. You may find this option useful if you already use MDL applications to process elements during plotting.
Don't Display Element	The element is not plotted. No MDL applications that desire to process the element are afforded the opportunity. When chosen, the other Output Actions controls are dimmed (disabled).
Call BASIC Macro Function	After the other Output Action controls have been applied to the selected element, additional processing is performed by the designated Function in the designated BASIC macro. You should not select Call BASIC Macro Function until your macro is fully prepared. This means you must have already used the BASIC Editor to debug the macro and to define a Pen Table Plot Element Hook entry point in your macro. See "Macro File" on page 11-22.

Priority

This setting lets you determines the order in which elements are plotted. The valid range for setting priority is -2147483648 to 2147483647. Priority of elements is determined by the following rules:

- Elements with a lower priority value are plotted before elements with a higher priority value.

- Unprioritized elements are always plotted before all prioritized elements.

To make an element meeting certain criteria plot before any other elements, you must prioritize *all* other elements (those that you

do not want plotted first) in the entire design. To make elements meeting certain criteria plot after other elements, you need only prioritize the elements you wish to be plotted last.

Do not prioritize elements unless it is significant to the plot, since prioritized elements require additional processing time and memory.

Fill

This setting lets you control whether or not filled areas are plotted for the selected filled elements, such as text, ellipses, shapes and complex shapes. When this setting is turned on, its option menu lets you choose between:

- On — all selected elements are plotted as filled.

- Off — all selected elements are plotted without fill.

When the Fill setting is turned off, the selected filled elements are plotted as they appear in the design file.

Color

This setting lets you control the color of the plotted output from the selected element. When this setting is enabled, you can select the required color by keying in its value or you can select the color graphically by clicking on the adjacent color tile. This opens a color picker dialog that lets you click on the required color. Selected elements then will be plotted in this color.

Fill Color

This setting lets you control the color of the plotted output of filled areas for the selected elements. When this setting is enabled, you can select the required color by keying in its value or you can select the color graphically by clicking on the adjacent color tile. This opens a color picker dialog that lets you click on the required color. Selected elements then will be plotted with the defined fill color.

Weight

This setting lets you control the line weight of the plotted output from the selected elements. When this setting is enabled, you can select the required line weight by keying in its value. Alternatively, you can select the weight graphically by clicking on the option menu and selecting the required weight value.

Style

This setting lets you control the line style and the scale of the selected elements. When this setting is enabled, you can key in

the required line style or click the Browse button to select the line style via a dialog box. When you choose a custom line style, you can use the Style Scale field to key in a scale value.

Macro File

This setting becomes active only when you have chosen the Master Control option Call BASIC Macro Function. It indicates that you wish the element to be passed to a function in a MicroStation BASIC macro as an `MbeElement`. When you choose the Call BASIC Macro Function option, the Macro File button is automatically activated, prompting you to identify the macro source file and the function within.

See "Using Macros to Specify Pen Table Output Actions" in Chapter 4 in the *MicroStation BASIC Guide* for more information about defining BASIC Program Entry Points and debugging Pen Table BASIC programs.

If color, weight or style resymbolization is specified in the Output Actions group box of the Modify Pen Table dialog box and the section also has a BASIC function associated with it, then the specified color, weight and style resymbolizations are applied to the `MbeElement` object by the time it is passed to the BASIC function. As always, the BASIC function is then free to manipulate the element, this would include overriding the color, weight and style.

In contrast to the above, if you assign an element priority it takes precedence over any priority assigned by a BASIC function. This exception allows pen tables to execute more efficiently.

Unloading the pen table

To disable pen table processing, you simply unload the pen table.

➤ **To unload the pen table**

1. From the Plot dialog box's PenTable menu, choose Unload.

or

From the Modify Pen Table dialog box's File menu, choose Exit/Unload.

12 Tutorial: Plot Resymbolization Using Pen Tables

In this tutorial, you will be shown how to perform plot resymbolization with a pen table. This will be done with simple exercises that show you the principles of creating different plotted output from the same design file, without changing the design itself.

Introduction

Plot resymbolization lets you create plotted output of a design with the symbology of some or all of the elements changed. These changes may be to make a drawing more clear or to make hard to see colors more visible. For example, your office standard may be to use colors such as white, yellow and orange against a black screen background. When plotted on a white sheet of paper, white elements automatically are plotted as black. The other elements, however, plot at their original colors. This may make them hard to see. Using a pen table, you can change the colors of these elements in the plot without affecting the original colors in the design file.

Where filled elements are used in a design, the order in which they were placed controls the final appearance on screen. That is, which overlapping filled areas obscure others. Using a pen table you can change the order in which these filled elements are plotted and control which overlapping elements are obscured.

Prepare for the exercises

In the following exercises you are introduced to plot resymbolization by working with simple design files. The procedures learned can be applied equally to your more complicated design files.

➤ **Open the sample design file**

1. Open the design file "resym1.dgn," which is located in the "…\Workspace\projects\tutorials\intro\dgn" directory.

   This design file has been saved with Views 1-4 open. The active level is set at 1 with no other levels displayed.

2. Turn off Views 2-4, leaving only View 1 open.

3. Use the *Fit View* view control to fit View 1.

View 1 displays a simple title block, consisting of a white rectangle with text in the lower right corner.

➤ **Open the plot dialog box and select a plotter**

1. From the File menu, choose Print/Plot.

The Plot dialog box opens.

2. In the Plot dialog box, click the Plotter Driver icon.

The Select Plotter Driver File dialog box opens.

3. From the list, select "psc650c.plt."

This is a color postscript printer. Make sure that you don't select "ps650c.plt" by mistake as this is a monochrome printer.

4. Click OK.
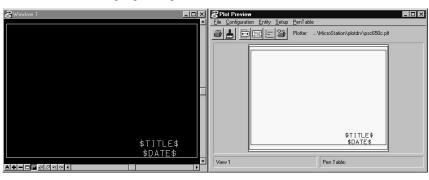
5. In the Plot dialog box, click the Preview Refresh icon.

The plot preview window opens and the name of the dialog box changes to Plot Preview.

6. Reposition and resize the Plot Preview dialog box so that it is side by side with View 1.

This lets you see both the design file elements and the proposed plotted output.

7. In the Plot Preview dialog box, click the Page Setup icon.

The Page Setup dialog box opens.

8. Turn on Rotate 90°.

9. Click OK.

The preview window updates to show the new orientation of the proposed plot.

View 1 and the Plot Preview dialog box set up for the exercises.

This file setup is used in the following exercises, which demonstrate features available with plot resymbolization.

➤ Create a pen table

1. From the Plot Preview dialog box's Pen Table menu, choose New.

The Create Pen Table file dialog box opens.

2. In the Files field, key in **resym1**.

The suffix ".tbl" automatically is added to the file name.

3. Click OK.

The Create Pen Table file dialog box closes and the Modify Pen Table dialog box opens. In the Processing Order list box of the Sections group box, a default section "NEW" is listed.

Using the Modify Pen Table dialog box, you will create a pen table that will resymbolize the example design.

Text Substitutions

You can use Text Substitutions to replace selected text in the design either with a special variable or with text of your choosing. In this example design, there are two text strings, "$DATE$" and "$TITLE$." The following exercise will replace these with the system date and a nominated title. Text substitution is a global action. That is, it applies to the entire design file not only to text elements of a particular color or on a particular level. It is recommended that when you place text that is to be replaced, you add a character, such as "$," at each end to distinguish it from other text in the design.

➤ Substitute the system date for $DATE$

1. In the Modify Pen Table dialog box click the Text Substitutions button.

The Text Substitutions dialog box opens.

2. From the dialog box's Edit menu, choose Insert Date.

Entries are inserted automatically for the Actual and Replacement fields. The entry also is inserted in the list box of substitutions.
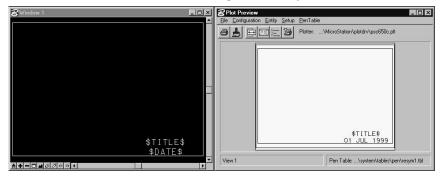
3. In the Plot Preview dialog box, click the Preview Refresh button.

The text string $DATE$ now is replaced with the current system date. This has no effect on the design file element, which remains as it was.

✍ If you had a text string other than $DATE$ in your design, you would have to change the Actual value to match that of your text string. Note also that the field is case sensitive — the Actual string must match that in the design file exactly.

Plot Preview shows the system date substituted for the text in the design file.

Similarly, you can replace other text in the design file with your own nominated text, as follows:

➤ **Substitute your title in the plotted output**

1. From the Text Substitutions dialog box's Edit menu, choose Insert New.

A new entry is placed in the list box, with a suggested Actual value of "Original."

2. In the Actual field, replace the word "Original" with "$TITLE$."

Make sure that you type it in exactly as it appears in the design file as the field is case sensitive.

3. In the Replacement field, key in "Sample Title."

4. Close the Text Substitutions window.

5. In the Plot Preview dialog box, click the Preview Refresh button.

The text string $TITLE$ now is replaced with the new text. Again, this has no effect on the design file element, which remains as it was.

The Plot Preview shows the replacement text.

Currently, your pen table is in memory only and would be lost if you exited MicroStation. You should save it to disk before continuing.

➤ **Save the pen table to disk**

1. From the Modify Pen Table dialog box's File menu, choose Save.

The pen table is saved. In the status bar, the following message appears: Pen Table <…\tables\pen\resym1.tbl> written.

Element Resymbolization

As well as text, you can substitute or resymbolize elements in the plotted output of a design file. This does not alter the design file elements. It affects only the plotted output.

➤ **Prepare for the exercise**

1. Turn on level 2 in View 1.

Level 2 contains several blocks and circles.

2. In the Plot Preview dialog box, click the Preview Refresh icon.

The elements appear in the preview window.

While the yellow elements are quite visible against the black background in View 1, against the white background of the

preview window they are hard to see. White elements, such as the title block, are plotted in black. Elements drawn in other colors retain these colors in the plotted output. This preview window shows us how the elements will appear when plotted on a white paper background. Using the pen table, you can change the color of the yellow elements when they are plotted.

➤ **Create a section to resymbolize the yellow elements as blue**

1. From the Modify Pen Table dialog box's edit menu, choose Rename Section.

   or

   In the Processing Order list box, double-click the entry "NEW."

   The Rename Section dialog box opens.

2. In the Name field, replace NEW with "Yellow to Blue."

3. Click OK.

   The Rename Section dialog box closes and the section is renamed in the Processing Order list box.

Next, you will define the elements that will be selected by this section. In this case, you want to select all yellow elements.

➤ **Set the Element Criteria**

1. If necessary, in the Modify Pen Table dialog box, click the Element Criteria tab.

2. In the Element Criteria folder, click the Color button.

   The Select Colors dialog box opens.

3. Click on the top yellow button.

4. Click OK.

   The Select Colors dialog box closes and the number 4 appears in the Color field.

✎ If you know the number(s) of the color(s) that you want selected, you can key in the number(s) directly in the Color field.

With the element selection criteria defined, you now define the output action. That is, change the color of the plotted element to blue.

➤ Define the output action

1. In the Modify Pen Table dialog box, click the Output Actions tab.

2. Turn on Color and in its field, key in **1**.

   The color tile turns to blue. You can also click on this tile and select the color graphically.

3. In the Plot Preview dialog box, click the Preview Refresh button.

   The yellow elements now appear in blue. No change is made to the elements in View 1, they remain yellow.

4. From the Modify Pen Table dialog box's File menu, choose Save.

   This saves our latest addition to the pen table.

Pen tables give you many other options for controlling plotted output. For example, you can turn on fill for shapes and/or ellipses. This can be done whether or not they appear filled in the design file. First, create a new section to plot filled shapes.

➤ Create a new section

1. From the Modify Pen Table dialog box's Edit menu, choose Insert New Section Above.

   The Insert Section dialog box opens, with the default name "NEW."

2. In the Name field, replace NEW with **Filled shapes and ellipses**.

3. Click OK.

   The new section appears in the Processing Order list box.

As before, you now define the section. First, you set the element selection criteria and then define the required output actions.

➤ Define the Section

1. In the Modify Pen Table dialog box, click the Element Criteria tab.

2. In the Type list box, scroll through to find Ellipse, then select it with a data point.

   The Ellipse entry highlights.

3. If necessary, scroll the list further to find Shape.

4. While holding down the <Ctrl> key, select Shape with a data point.
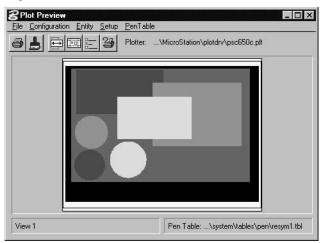
The Shape entry highlights. Because you held down the <Ctrl> key as you selected Shape, the previous selection (Ellipse) remains selected.

5. Click the Output Actions tab.

6. Turn on Fill and from its option menu, choose On.

7. From the dialog box's File menu, choose Saved.

It is a good idea to save your pen table after each section is defined. This ensures that the information is saved to disk.

8. In the Plot Preview dialog box, click the Preview Refresh icon.

The preview window updates, displaying all the shapes and ellipses as filled elements.

Plot preview now shows all shapes and ellipses as filled.

Notice that the preview shows that the filled shapes include the title block (black) and the violet rectangle, which appears as a dashed line in the design file.

Notice also that the two yellow elements are shown again as yellow. The section that changes them to blue has been ignored. This illustrates a very important point. Any elements that have been previously modified for the output cannot be modified by subsequent sections. This can be demonstrated very simply as follows:
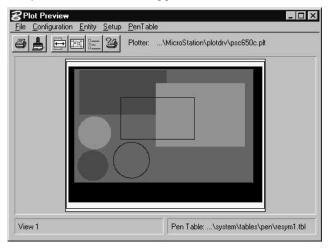
➤ **Change the Processing Order**

1. In the Processing Order list box, select the section "Yellow_to_Blue."

2. Click the Up button below the list box.

   The selected section is moved up above the previous section.

3. In the Plot Preview dialog box, click the Preview Refresh icon.

   The yellow elements now appear as blue but not filled.

To make the blue elements appear filled, you have to change the output action settings to include these instructions. Do that now.

➤ **Modify the Yellow_to_Blue section**

1. If necessary, select the section "Yellow_to_Blue."

2. Click the Output Actions tab.

3. Turn on Fill and from its option menu, choose On.

4. Turn on Fill Color and in its field, key in **1**.

5. From the dialog box's File menu, choose Save.

6. In the Plot Preview dialog box, click the Preview Refresh icon.

   The preview window updates, showing the yellow elements as blue filled elements.

Next, you will turn off fill for the title block and for the rectangle represented by the violet dashed line in the design file. In order

to do this, you must take into account where to position the section in the Processing Order. In the current pen table, the section "Filled_shapes_and_ellipses" acts on shapes and ellipses. If you place the section after this entry, it will be ignored. Therefore, you will place the entry above "Filled_shapes_and_ellipses."

➤ **Create a new entry to plot title block and dashed rectangle unfilled**

1. In the Processing Order list, select "Filled_shapes_and_ellipses."

2. From the Modify Pen Table dialog box's Edit menu, choose Insert New Section Above.

   The Insert Section dialog box opens, with the default name NEW.

3. In the Name field, replace NEW with **Unfilled shapes**.

4. Click OK.

   In the Processing Order list, the new entry is inserted above "Filled_shapes_and_ellipses."

➤ **Define the Element Criteria**

1. Click the Element Criteria tab.

2. In the Type list box, select Shape.

3. Click the Color button.

   The Select Colors dialog box opens.

4. In the Select Colors dialog box, click the White and Violet buttons (first and sixth buttons in the top line).

5. Click OK.

   The dialog box closes and the colors' numbers (0,5) appear in the Color field.
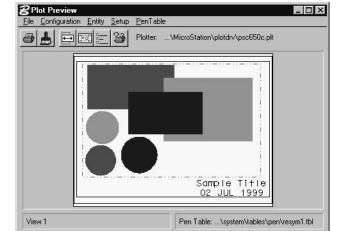
➤ **Define the Output Actions**

1. Click the Output Actions tab.

2. Turn on Fill and from its option menu, choose Off.

3. From the Modify Pen Table dialog box's File menu, choose Save.

4. In the Plot Preview dialog box, click the Preview Refresh icon.

The preview window updates showing the title block and dashed rectangle as unfilled elements. All other shapes and ellipses still are filled.

The title block and dashed rectangles now are displayed unfilled.

If you delete a section, then the plotted output reverts to the default setting. To see this in action, delete the section that changes the color of yellow elements to blue.

➤ **Delete a section**

1. In the Processing Order list box, select "Yellow_to_Blue."

2. From the Modify Pen Table dialog box's Edit menu, choose Delete Section.

The section is deleted from the Processing Order list.

3. In the Plot Preview dialog box, click the Preview Refresh icon.

The blue elements are returned to the original design file color of yellow.

In this sample file, some of the rectangles are overlapped by others. This is most obvious when filled shapes are plotted. The order in which they are displayed is controlled by the order in which they were placed in the design file. That is, the element that was placed later overlaps those that were placed previously. In this design file, the red block was placed before the green block, which in turn was placed prior to the yellow block. You can change this order for plotting purposes so that a different color block appears "on top." You do this by applying a priority

setting to the elements. When creating a plot using a pen table, elements that have no priority setting are processed before those with a priority setting.

For this example, you will assign priority settings to the red and green blocks such that the red block will be on top and the green block between the red and yellow blocks. Because the yellow block is to be on the bottom, it does not have to be prioritized. Any elements that have no priority setting are plotted before those that have a priority assigned to them. Again, you will have to be aware of the Processing Order. The new sections have to be placed prior to the "Filled_shapes_and_ellipses" section.

When giving elements priority settings it is good practice to assign values that allow you room to insert other entries at a later date. For example, in this exercise you will assign the red shape a priority of 100 and the green shape a priority of 50.

➤ **Create two new sections**

1. In the Processing Order list, select "Filled_shapes_and_ellipses."

2. From the Modify Pen Table dialog box's Edit menu, choose Insert New Section Above.

   The Insert Section dialog box opens with the default name NEW.

3. In the Name field, replace NEW with **Red shape priority**.

4. Click OK.

   In the Processing Order list, the new entry is inserted above "Filled_shapes_and_ellipses."

5. Repeat steps 2 through 4 naming the section **Green shape priority**.

➤ **Define the Element Criteria**

1. With the "Green_shape_priority section" selected, click the Element Criteria tab.

2. In the Type list box, select Shape.

3. Click the Color button.

   The Select Colors dialog box opens.

4. In the Select Colors dialog box, click the top green button.

5. Click OK.

   The dialog box closes and the color's number (2) appears in the Color field.

6. In the Processing Order, select "Red_shape_priority."

7. In the Type list box, select Shape.

8. Click the Color button.

    The Select Colors dialog box opens.

9. In the Select Colors dialog box, click the top red button.

10. Click OK.

    The dialog box closes and the color's number (3) appears in the Color field.

➤ **Define the Output Actions**
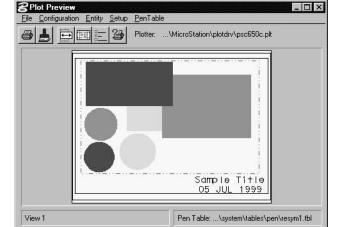
1. With the "Red_shape_priority" section selected, click the Output Actions tab.

2. Turn on Fill and from its option menu, choose On.

3. Turn on Priority and in its field, key in **100**.

4. In the Processing Order list, select "Green_shape_priority."

5. Turn on Fill and from its option menu, choose On.

6. Turn on Priority and in its field, key in **50**.

7. From the Modify Pen Table dialog box's File menu, choose Save.

8. In the Plot Preview dialog box, click the Preview Refresh icon.

The preview window updates showing the red rectangle on top with the green rectangle between the red and yellow rectangles.

The red rectangle now is on top, with the green rectangle between it and the yellow rectangle.

With the green and red shapes given priorities of 50 and 100 respectively, you have the option of positioning other shapes between them. For example, if you gave yellow shapes a priority such as 75, the yellow rectangle would be plotted on top of the green and below the red rectangle.

➤ **Assign the yellow rectangle a priority of 75**

1. In the Processing Order list, select "Filled_shapes_and_ellipses."

2. From the Modify Pen Table dialog box's Edit menu, choose Insert New Section Above.

The Insert Section dialog box opens, with the default name NEW.

3. In the Name field, replace NEW with **Yellow shape priority**.

4. Click OK.

In the Processing Order list, the new entry is inserted above "Filled_shapes_and_ellipses."

➤ **Set the Element Criteria and Output Actions for the new section**

1. With the "Yellow_shape_priority" section selected, click the Element Criteria tab.

2. In the Type list, select Shape.

3. Click the Color button.

   The Select Colors dialog box opens.

4. In the Select Colors dialog box, click the top yellow button.

5. Click OK.
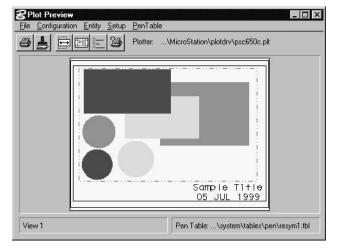
   The dialog box closes and the color's number (4) appears in the Color field.

6. Click the Output Actions tab.

7. Turn on Fill and from its option menu, choose On.

8. Turn on Priority and in its field, key in **75**.

9. From the Modify Pen Table dialog box's File menu, choose Save.

10. In the Plot Preview dialog box, click the Preview Refresh icon.

   The preview window updates still showing the red rectangle on top but now with the yellow rectangle between the red and green rectangles.

The yellow rectangle now appears between the red and green rectangles.

Notice that it does not matter in what order the priority sections are placed, the elements are plotted in the correct order relative to their priority settings.

That concludes this introduction to plot resymbolization with pen tables.

13 Plotting Using Image Manager

This section describes how you can use plot or print with Image Manager.

Overview

MicroStation's Image Manager is an application that lets you control the display of one or more image files within a MicroStation design file view. It is an optional application delivered with MicroStation. When installed, each view in a design file may be used to display image files. This lets you save up to eight separate groups of images in one design file.

Information on the image files is stored with the design file. As well, this information may be saved separately as a project file (with a ".prj" extension), letting you open it in other design files. Project files contain basic parameters about your project and working session, as well as information that defines the relationship between the image files.

MicroStation's Image Manager supports PostScript and HPGL2/RTL devices, as well as all raster devices supported through the Windows Print Manager.

In a typical Image Manager project where aerial photography is used, a design file view contains a mosaic of images depicting the region of interest. The mosaic can include an unlimited number of images each referenced to the vector data contained in the active design file and any reference files attached to it. Similarly, using Image Manager, you can assemble a group of rendered images for plotting.

Whether you plot to a color or grey scale postscript device, images from various sources can be assembled together and the colors for each image are represented correctly on the output. With this flexibility, you can integrate black and white scanned documents, gray scale or color aerial photographs, transparent monochrome bitmap layers, satellite imagery and design files to

produce a full color document on color devices or a gray scale document on gray scale devices.

MicroStation's Image Manager lets you produce hybrid plots of raster images along with raster vector data and design vector data. To produce such a hybrid plot for a view, choose Print/Plot from the Image Manager File menu.

Plotting devices for Image Manager

MicroStation's Image Manager lets you plot to the following devices:

- PostScript plotters

- HPGL2/RTL plotters

- other supported plotters ("generic" plotters)

- your system printer

MicroStation's plot driver file creates the vector data portion of the plotfile while Image Manager's driver creates the raster data portions. The resulting composite plotfile includes information about both the vector data and raster data of your image.

Every supported plotting device has a plot driver file with file extension ".plt," which controls the plotting process. Your system's plot file is "printer.plt." Additionally, MicroStation provides three suffix files. Each suffix file consists of a set of MicroStation plot commands suitable for a type of plotter. The suffix files are as follows:

- `imps.suf` commands for PostScript plotters (DOS/Windows).

- `imrtl.suf` commands for RTL plotters, (DOS/Windows).

- `imgen.suf` commands for other MicroStation-supported plotters and for your system printer (Windows only).

Each Image Manager usable plot driver consists of a standard plot driver file to which the appropriate set of MicroStation plot commands have been added in a suffix file (".suf" extension). The resulting file is a concatenated plot driver (with ".plt" extension).

Four sample concatenated plot drivers are delivered with MicroStation, as described in the following table.

Standard MicroStation Plot Driver	Type of plot driver	Suffix file	Concatenated plot driver
HP650c.plt	RTL	imrtl.suf	**imrtl.plt**
pscript.plt	postscript	imps.suf	**imps.plt**
pscriptc.plt	postscript color	imps.suf	**impsc.plt**
printer.plt	PC printer	imgen.suf	**imgen.plt**

Concatenated plot driver files that use either `imrtl.suf` or `imps.suf` should not be configured to plot directly to a port (for example, by defining `default_outfile=<port or printer>`). This is because they post-process the plotfile generated by the standard plotting code. That is, the standard plotting code produces a plot with no Image Manager raster data. The MDL application referenced in the ".suf" file then processes this raster data and merges it into the existing plotfile. If there is no original plotfile with which to merge, then the resulting plot contains no Image Manager raster data.

Only the concatenated plot driver file "imgen.plt" processes Image Manager raster along with the vector data. Therefore, you can plot directly to a port or printer when using this plot driver file.

Image Manager Plotting dialog boxes

For each of the concatenated plotter drivers, there is an associated plotting dialog box. When you select *Setup > Devices* in the Plot dialog box, the MicroStation Plotting dialog box that opens is dependent on the current plotter driver (.plt file), as shown in the following table.

PLT File	Associated MicroStation Plotting dialog
imgen.plt	Image Manager - Image Plotting
imps.plt impsc.plt	Image Manager - PostScript Image Plotting
imrtl.plt	Image Manager - HPGL2/RTL Image Plotting

Image Manager - Image Plotting dialog box

Used with "imgen.plt" plotter driver, it is a concatenation of "printer.plt" and "imgen.suf."

Image Manager - Image Plotting dialog box
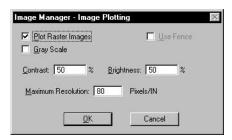
Image Manager - PostScript Image Plotting dialog box

Used with either "imps.plt" or "impsc.plt" plotter drivers, which are a concatenation of "pscript.plt" and "pscriptc.plt" respectively, and "imps.suf."
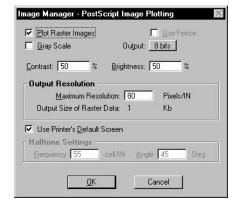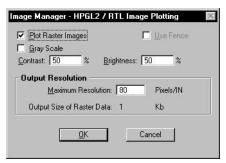
Image Manager - PostScript Image Plotting dialog box

Image Manager - HPGL2/RTL Image Plotting dialog box

Used with "imrtl.plt" plotter driver, which is a concatenation of "hp650c.plt" and "imrtl.suf."

Image Manager - HPGL2 / RTL Image Plotting dialog box

Plotting dialog box(es) settings

Settings in the various plotting dialog boxes let you control the output to the respective plotters or printers.

Plot Raster Images (default ON)

Turn off Plot Raster Images to disable the plotting of Raster images.

✍ This parameter is saved with a project.

Gray Scale (default OFF)

- RTL — Turn on Gray Scale to plot images in gray scale RTL format even if they are color images.

  If you wish to print color images and your RTL device supports color, leave Gray Scale turned off.

- PostScript — If you turn on Gray Scale, images are plotted with a half-toning technique that renders shades of gray.

  Leave Gray Scale turned off to plot images in color PostScript format.

For both RTL and PostScript devices, the following also applies:

- When plotting to a black and white RTL or PostScript device, you should turn on Gray Scale, even for color images. This saves time and minimizes the disk space required to plot the image.

- Color mosaics of multiple images can be printed correctly even though they may not display correctly on your monitor. For example, if you have an 8 bit display and one image is

active, all images use its color table for on-screen display.
However, during plotting, each image in a view is plotted to a
color PostScript device using its own color table like with a 24
bit display.

- The on/off status of the Gray Scale option does not affect the
  design files. Therefore, you can print color design elements in
  a color PostScript format even if the Gray Scale check button
  is on (to plot the images in gray scale).

✍ This parameter is saved with a project.

Use Fence (default OFF)

Used to indicate whether the raster images should be clipped to
the fence when plotting the view. The Use Fence option is only
available when a fence is active.

✍ This parameter is saved with a project.

Output

Select from 8 or 24 bit output. Some PostScript printers are not
capable of handling more than one 8 bit color table for generating
a printout. This means that several images with different colors
would print using the same color table. When this happens, select
24 bit. Each image is then printed using its own color table, thus
producing the desired output.

Contrast and Brightness

Used to set the contrast and the brightness of images in your
plots. These settings apply only to images when plotting. They do
not affect the on-screen display of images.

Output Resolution

It is recommended that you select the maximum possible number
of pixels per inch for the output resolution. You can use a lower
resolution to obtain a quick draft output but the higher resolution
should be used to obtain high quality output.

Maximum Resolution

To ensure maximum quality for your plot, you should set this
value to the same resolution as that of your printing/plotting
device. With devices that use half-toning, however, the maximum
resolution that you can achieve is the net resolution after half-
toning (that is, the net resolution will be less that the overall
resolution of the device). Since a value set higher than necessary

usually does not provide added resolution, it is advisable to determine the optimum value to minimize the size of your plot file.

Calculating the optimum output resolution

To calculate the output resolution that will produce the smallest plot file size:

1. Determine the net resolution of your device. This value should be less than the resolution of the printer.

   The Image Manager default value is 80, since it is the typical net resolution for 300 dpi printers that plot in gray scale with half-toning.

2. Optimize your plot file size by determining whether your plot requires the full available resolution.

Use the following formula to optimize the plot file size:

(Image Width/Pixel Size) / Width of Plot = Optimal Resolution.

For example, if the image width is 1000 meters, the pixel size is 1 meter and the width of the plot is 5 inches, then:

(1000/1) / 5 = 200 dpi

✎ Use the Image Location Editor dialog box to find the pixel size and the image dimensions (select File > Properties > Location in the Image Manager dialog box).

In this example, the maximum quality output can be achieved with a resolution set at 200 dpi. Consider also:

- Any higher value creates a larger plot file but does not increase quality.

- If the net (true) resolution of your device is lower than 200 dpi, then any value higher than the true resolution also creates a larger plot file but does not increase quality.

   In this instance, if your device does not contain the true resolution necessary to achieve the maximum possible resolution inherent in your image, then your plot quality cannot be maximized.

✎ This parameter is saved with a project.

After the Maximum Resolution is set, the Output Size of Raster Data field displays the amount of raster data that will be created. The total size of the PostScript file also includes the vector data.

Use Printer's Default Screen (default ON)

Turn off only when you need to override the standard screen settings of your PostScript device.

Usually, the default screen settings of your printer are optimal for your particular device. Turn off *Use Printer's Default Screen* only if you have experienced a problem with the quality of your plot (such as dark images).

✍ This parameter is saved with a project.

Halftone Settings

These settings are enabled when Use Printer's Default Screen is turned off. They are used to set the frequency and Angle of the screen on the half-toning printer.

Frequency (default 55)

This setting, in cells per inch, determines the interval between the centers of the halftone dots without regard to the dot size. The higher you set the frequency, the smoother are the apparent changes in tone. If you set the frequency too high for your printer, however, you will lose image contrast and the result will be darker than expected.

The default value for Frequency is 55, which should provide a near-optimum result on a 300 dpi printer. For a lower resolution printer, use a lower Frequency value (around 30 to 50 cells per inch). For a higher resolution printer, use a higher value (80 cells per inch or more).

✍ This parameter is saved with a project.

Angle (default 45)

Used to set the angle of the half-toning screen of your printer. The Angle value determines the number of degrees by which the halftone screen rotates in relation to the device coordinate system.

✍ This parameter is saved with a project.

OK

Accepts the values in the dialog box and closes it.

Cancel

Closes the dialog box without changing the parameters.

Plotting images

Plotting images from Image Manager is similar to standard plotting, except that only particular plotter driver files can be used.

➤ **To plot from Image Manager**

1. From the Image Manager File menu, select Print/Plot.

   The Plot dialog box opens.

2. From the Setup menu, choose Driver.

   The Select Plotter Driver File dialog box opens.

3. Select a concatenated plot driver suitable for your plotter or your system printer.

4. Click OK.

   The Select Plotter Driver File dialog box closes and the selected plot driver displays in the Plotter field.

5. From the Setup menu, choose Device.

   The relevant Plotting device dialog box opens.

6. Make any necessary changes to the various fields in the dialog box.

7. Click OK.

   The plotting device dialog box closes.

8. In the Plot dialog box, click the Plot icon.

   or

   From the Plot dialog box's File menu, choose Plot.

   If you are outputting to the system printer, when you select Plot, the image file is processed and then printed. If you are outputting to a plotter other than the system printer, the Save Plot As dialog box opens.

9. Choose the name and location for the output plotfile.

10. Click OK.

    The plotfile is created on disk.

When the system printer is the output device, the image is plotted immediately. When your output is to a plotter, the plotfile is saved to disk. To plot the (saved) plotfile, requires that you complete one more step.

➤ **To plot out your plot files**

1. Open an MS-DOS window.

2. Change the directory to where the plot file is located.

3. Use the key-in:

`COPY /B <filename> <printer_port>`

Where `<filename>` is the name of the plot file and `<printer_port>` is the port to which the printer/plotter is attached (such as *lpt1:*)

Plotting begins when the plotter receives the plotfile (containing the image according to the selected parameters).

Additional Information About Plot Configuration Files

Before you try to plot with Image Manager, the appropriate statements must be present in the plot configuration file. If the appropriate statement(s) are not present in your configuration, the device option from the setup menu will be grayed.

Plotting Design 87 and 88 Elements

Key-ins can be used to enable or disable the plotting of Design raster element types 87 and 88 when using the Plot feature.

To turn on this feature:

Key-in: IMGMANAGER PLOT8788 SETON

When the toggle is set to ON, MicroStation plots type 87 and 88 elements with the same options used to print raster files (clipping to the fence extent, converting to gray scale, and so forth).

To turn off this feature:

Key-in: IMGMANAGER PLOT8788 SETOFF

When the toggle is set to OFF, MicroStation ignores type 87 and 88 elements.

RTL Plotting - Exceeding Installed Memory on Plotter

If you are not using the Print Manager to plot files, it may happen that the raster portion of the plot does not fit entirely into the

plotter's memory. When this happens, several options are available:

• Add more memory to the plotter.

• Reduce the size of the plot file by reducing the resolution or paper size.

• Use the Plot On The Fly Feature.

The Plot On The Fly feature is activated by inserting a special keyword into your plt file. When this keyword is found, MicroStation generates the plot file in such way that when the plotter's memory is full, it:

• Stops input of data.

• Starts the drawing.

• Starts input and plotting of data one band at a time until it reaches the end of the file.

This method is slower but allows plotting large raster files on a plotter with minimal memory installed.

The only condition to use this feature is for the design file to entirely fit into the plotter's memory. Conventional ways require the raster data be spooled first and then the vector data (design file). When using Plot On The Fly, the vector data is spooled first. This method ensures that no matter how large the raster portion is, it will never outrun the plotter's memory.

Using Plot On The Fly always produces translucent vectors. Vector information will be transparent and some colors, like yellow, may not show properly when an aerial photograph is beneath the vector data.

➤ **To modify the PLT file for Plot On The Fly**

1. Using a text editor, open the plt file you will use to generate the plot file.

2. Begin a new line anywhere and insert the following:

   RTL_PLOT_ON_THE_FLY=1

3. Save and exit.

➤ **To send the plot file to the plotter**

1. Open an MS-DOS window.

2. Change the directory to where the plot file is located.

3. Type the following command:

```
mode lpt1 retry=p
```

This instructs the copy command to try sending data to the printer until it responds. The printer will not respond until it has completed processing the data from its memory.

4. Send your file to the plotter by typing:

```
copy /b <filename> <printer_port>
```

Where `<filename>` is the name of the plot file and `<printer_port>` is the port to which the printer/plotter is attached (such as *lpt1:*)

The prompt returns when the copy command finishes copying the plot file to the printer port.

This can take several minutes to several hours depending on the amount of raster data remaining to be plotted when the Plot On The Fly mode begins. Your computer will be tied up for the duration of the process. Remember, the remaining raster data is spooled one band at a time.

Appendix A: Plotting Key-ins

This chapter describes the key-ins that are available for plotting functions.

Plotting key-ins

Key-ins can be used to submit plots to a printer/plotter. They provide options for customizing your plot files to specific criteria. and can be used in BASIC macros or in MDL applications.

Plotting key-in commands are provided by the "plotcfg.ma" plotting application.

One-step commands

You can use one-step commands to set up and produce a plot in one step. The following table describes MicroStation one-step commands and, where applicable, the associated parameters. Plotting key-in commands are provided by the plotcfg.ma plotting application.

Key-ins	Description/Use
Plot File Creation Commands	
PLOT FENCE [output_filename]	• Selects an existing fence as the plot entity and produces a plot. • You must specify the output filename unless the active printer is the system printer. • This command forces a repaint of the preview area if visible. • Enter this complete key-in; do not abbreviate. This key-in is not case sensitive.

Key-ins	Description/Use
PLOT VIEW `<view_number>` `[output_filename]`	• Selects a view specified by number (1-8, inclusive) as the plot entity and produces a plot. • Specify output filename unless active printer is system printer. • If the specified view is off, it is turned on. • This command forces a repaint of the preview area if it is visible. • Enter this complete key-in; do not abbreviate. This key-in is not case sensitive.
PLOT `<configuration_filename>` `[output_filename]`	• Loads a specified plot configuration (.ini) file and produces a plot. It does nothing if the configuration file does not exist or is invalid (is not of the correct format or was created from a file whose dimension differs from that of the active design file). • Specify output filename unless active printer is system printer. • This command forces a repaint of the preview area if visible.

Combined Commands

Combined Commands are a series of key-in commands that you can use together to both set up and produce a plot. To use combined commands, you separate each one with a semi-colon (;).

These Combined Commands are grouped in the following categories:

- Plotter Selection Commands
- Plot Execution Commands
- Plot Preview Commands
- Entity Selection Commands
- Layout Commands
- Display Attributes Commands
- Border Commands
- Miscellaneous Commands
- Dialog Display Commands
- Paper/Page/Form Commands

Key-in	Description/Use
Plotter Selection Commands	
SET PLOTTER [plotter_name]	• If a plotter name (or plotter driver filename) is specified, selects that plotter as the active printer. • If the plotter name is omitted, displays the Select Plotter Driver File dialog box.
Plot Execution Command	
PLOT EXECUTE [output_filename]	• Produces a plot using existing settings. • Specify output filename optional if active printer is system printer. • Selecting a new plotter (loading a new plotter driver file) clears most settings. It is best to select a plotter, specify settings, and then use the plot execute command to produce the plot.
Plot Preview Commands	
PREVIEW	• Displays the Plot dialog box and its preview area, or expands the preview area if the Plot dialog box is already displayed. • If the Plot dialog box is already displayed with the preview area visible, this command does not force an update to synchronize the preview image with plotcfg's internal state. • Use "plot preview refresh" to synchronize the preview image if it is visible.
PLOT PREVIEW REFRESH	• Repaints the visible plot preview area.
Entity Selection Commands	
PLOT ENTITY FENCE	• Selects the fence, if any, as the active plot entity.
PLOT ENTITY VIEW <view_number>	• Selects a view specified by number (1 - 8, inclusive) as the plot entity. • If the specified view is off, it is turned on.

Key-in	Description/Use
PLOT ENTITY CONFIGURATION `[configuration_filename]`	• If a configuration file is specified, loads that file and makes it the plot entity. Otherwise, selects the most-recently loaded plot configuration, if any, as the current plot entity. • If necessary, searches the directories specified by the MS_PLOTINI configuration variable, to find the configuration file. • This command forces a repaint of the preview area if visible.
Layout Commands	
PLOT MARGINS BOTTOM `<value>` `[units_string]`	• Specifies the vertical plot margin (plot offset) and deselects auto-center and maximize modes. • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.
PLOT MARGINS LEFT `<value>` `[units_string]`	• Specifies the horizontal plot margin (plot offset) and deselects auto-center and maximize modes. • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.
PLOT MARGINS CENTER	• Centers the plot and enables auto-centering mode.
PLOT SCALE `<percent_scale%|working_unit_scale>`	• Sets the plot scale and recalculates the plot's horizontal and vertical sizes. • If the scale is specified as a percentage, the percentage is relative to the plot's normal (maximized) plot size. • If the scale is specified as a working unit value, the scale maps that number of working units to one plotter unit (the units specified in the plotter driver file resolution record).
PLOT SIZE HEIGHT `<value>` `[units_string]`	• Sets the vertical plot size, recalculates the horizontal size to maintain the plot's original aspect ratio, and deselects maximize mode. • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.

Key-in	Description/Use
PLOT SIZE WIDTH `<value>` `[units_string]`	• Sets the horizontal plot size, recalculates the vertical size to maintain the plot's original aspect ratio, and deselects maximize mode. • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.
PLOT SIZE MAXIMIZE	• Selects maximize mode and sets the horizontal and vertical plot size to values that maximize the plot size, without rotating the page or changing the plot's aspect ratio.

Display Attributes Commands	
PLOT OPTIONS CAMERA `[on/off]` PLOT OPTIONS CONSTRUCTIONS `[on/off]` PLOT OPTIONS DATAFIELDS `[on/off]` PLOT OPTIONS DIMENSIONS `[on/off]` PLOT OPTIONS FASTCELLS `[on/off]` PLOT OPTIONS FASTCURVES `[on/off]` PLOT OPTIONS FASTFONT `[on/off]` PLOT OPTIONS FASTREFCLIP `[on/off]` PLOT OPTIONS FENCEBOUNDARY `[on/off]` PLOT OPTIONS FILLS `[on/off]` PLOT OPTIONS LEVELSYMBOLOGY `[on/off]` PLOT OPTIONS LINESTYLES `[on/off]` PLOT OPTIONS LINEWEIGHTS `[on/off]` PLOT OPTIONS PATTERNS `[on/off]` PLOT OPTIONS REFBOUNDARIES `[on/off]` PLOT OPTIONS TAGS `[on/off]` PLOT OPTIONS TEXT `[on/off]` PLOT OPTIONS TEXTNODES `[on/off]`	• These commands enable or disable view attributes, similar to the toggle buttons in the Plot Options dialog box. The "plot option" commands (except for fenceboundary) have no effect unless a plot configuration file defines the plot entity. • On/Off — In commands accepting on or off, the keyword may be omitted, in which case on is assumed. • To turn the border display on or off, use plot border on/off. A plot options border command is not included.

Key-in	Description/Use
Border Commands	
PLOT BORDER PEN `<pen_number>`	• Specifies the pen (1 - 255, inclusive) used to draw the plot border (outline and text). • Border — Commands that specify plot border attributes such as pen, comment, filename on/off, and so forth do not automatically enable the border. They define attributes to be used if the border is plotted. If these attributes are turned off, the border will not plot even when enabled. • Use plot border [on/off] to enable or disable the border.
PLOT BORDER COMMENT `<string>`	• Specifies an arbitrary border comment string. The string is limited to 80 characters. • Border — Commands specifying plot border attributes do not automatically enable border, they define attributes if the border is plotted. If attributes are turned off, the border will not plot even when enabled. • Use plot border [on/off] to enable or disable border.
PLOT BORDER FILENAME `[on/off]`	• Enables or disables plotting of active design file name on plot border. • Border — Commands specifying plot border attributes do not automatically enable border. They define attributes if the border is plotted. If attributes are turned off, the border will not plot even when enabled. • Use plot border [on/off] to enable or disable border.
PLOT BORDER TIME `[on/off]`	• Enables or disables the plotting of the date and time on the plot border. • Border — Commands specifying plot border attributes do not automatically enable border. They define attributes if the border is plotted. If these attributes are turned off, the border will not plot even when enabled. • Use plot border [on/off] to enable or disable border.
PLOT BORDER WIDTH `<value>` `[units_string]`	• Specifies width of border outline in physical units. 0.0 is a valid border width, and means "use thinnest possible line." • Border — Commands specifying plot border attributes do not automatically enable border. They define attributes if the border is plotted. If these attributes are turned off, the border will not plot even when enabled. • Use plot border [on/off] to enable or disable the border. • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units defined by active plotter driver file's resolution record.

Key-in	Description/Use
PLOT BORDER TEXTHEIGHT `<value>` `[units_string]`	• Specifies height of any text appearing on plot border. • Border — Commands specifying plot border attributes do not automatically enable border. They define attributes if the border is plotted. If these attributes are turned off, the border will not plot even when enabled. • Use plot border [on/off] to enable or disable the border. • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.
PLOT BORDER `[on/off]`	• Enables or disables plot border as a whole.
Miscellaneous Commands	
PLOT PEN COLOR `<pen_number, r, g, b>`	• Specifies the RGB color for a specific pen (pen numbers between 1 and 255, inclusive) or for the plot background (pen number 0). • For standard pens, R, G, and B must be in the range 0 - 255, inclusive. • To select the background color defined in the master file's color table, you can use the special RGB value "-1, -1, -1." • Background colors are intended for use with plotter drivers that produce raster files such as JPEGs.
PLOT QUALITY `<value>`	• Specifies the plot quality/compression (0 - 100, inclusive) amount for files created by the JPEG raster output driver. Zero corresponds to low quality, high compression, while 100 corresponds to high quality, low compression.
Dialog Display Commands	
PLOT	• Displays the Plot dialog box, hiding the preview area if visible.
PLOT PRINTER `<printer name>`	• In Windows 95 and Windows NT platforms, use the "plot printer" key-in to select a particular local or network Windows printer as the system printer (if a plot driver specifying model = sysprinter is loaded, such as printer.plt).
PRINT	• Equivalent to plot.
Paper/Page/Form Commands	
PAGE SETUP	• Displays the Page Setup dialog box (or its system printer equivalent).

Key-in	Description/Use
PLOT PAPER HEIGHT `<value>` `[units_string]`	• Sets the vertical size of the currently selected paper size (form). • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.
PLOT PAPER WIDTH `<value>` `[units_string]`	• Sets horizontal size of the currently selected paper size (form). • Units — Valid unit strings are dots, mm, cm, dm, m, in, and ft. When string is omitted, assumes units are defined by the active plotter driver file resolution record.
PLOT PAPER ROTATE `<cw/ccw>`	• Rotates page or (plot) clockwise or counterclockwise by 90 degrees. • On Windows 95 and Windows NT platforms with a system printer selected, using either "cw" or "ccw" is equivalent to specifying a landscape orientation on the standard Windows print dialog box.
PLOT PAPER ROTATE `[on/off]`	• Sets the page rotation to 0 (off) or 90 (on) degrees. If the page is rotated 90 degrees, the direction specified by the plotter driver file's "rotate" record (or by the last plot paper rotate cw/ccw command) is used. • "on/off" — In commands accepting on or off, the keyword may be omitted, in which case on is assumed. • On Windows 95 or Windows NT platforms with a system printer selected, use "Rotate [off]" for portrait orientation or "Rotate [on]" for landscape orientation.
PLOT PAPER NAME `<paper_size_name>`	• Selects a new paper size (form). • On Windows 95 or Windows NT platforms with a system printer selected, use "plot paper name" to select paper size. • Note: You must select a paper size (form) name corresponding to a physical paper tray that the printer can use (see "Paper size (Form) names for Windows platforms" below).
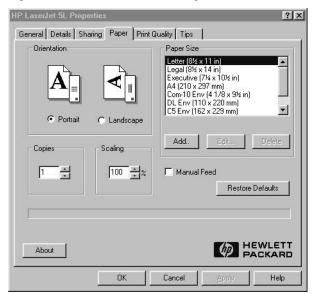
Paper size (Form) names for Windows platforms

Certain paper size (form) names are common for a majority of commonly used printers/plotter. However, many vendors now include special form names that are specific for each printer. To ensure that you specify a correct form name for your printer, you should check the names and sizes in the Document Defaults dialog box.

➤ **To view printer paper sizes in Windows 95**

1. From the Start menu's Settings sub-menu, choose Printers.

The Printers window opens displaying the installed printers.

2. Double-click the required printer's icon.

The selected printer's dialog box opens.

3. From the Printer menu, select Properties.

The printer's properties dialog box opens.

4. Click the Paper tab.

The Paper tab displays settings for paper sizes, including legal Paper Size names for the selected printer.
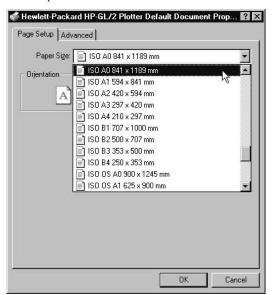
Windows 95 printer properties dialog box, showing Paper Size settings for LaserJet 5L.

➤ **To view printer paper sizes in Windows NT**

1. From the Start menu's Settings sub-menu, choose Printers.

The Printers window opens, displaying the installed printers.

2. Double-click the required printer's icon.

The selected printer's dialog box opens.

3. From the Printer menu, select Document Defaults.

The printer's Document Properties dialog box opens.

4. Open the Paper Size option menu to view paper size names for the printer.

Typical Paper Size option menu for HP-GL/2 plotter in Windows NT.

Typical Paper Size (Form) Names for Windows 95 and Windows NT Platforms	Description
LETTER	Letter 8 1/2 x 11 in
LETTERSMALL	Letter Small 8 1/2 x 11 in
TABLOID	Tabloid 11 x 17 in
LEDGER	Ledger 17 x 11 in
LEGAL	Legal 8 1/2 x 14 in
STATEMENT	Statement 5 1/2 x 8 1/2 in
EXECUTIVE 10 1/2 in	Executive 7 1/4 x
A3	A3 297 x 420 mm
A4	A4 210 x 297 mm
A4SMALL	A4 Small 210 x 297 mm

Typical Paper Size (Form) Names for Windows 95 and Windows NT Platforms	Description
A5	A5 148 x 210 mm
B4	B4 (JIS) 250 x 354
B5	B5 (JIS) 182 x 257 mm
FOLIO	Folio 8 1/2 x 13 in
QUARTO	Quarto 215 x 275 mm
10X14	10x14 in
11X17	11x17 in
NOTE	Note 8 1/2 x 11 in
ENV_9	Envelope #9 3 7/8 x 8 7/8
ENV_10	Envelope #10 4 1/8 x 9 1/2
ENV_11	Envelope #11 4 1/2 x 10 3/8
ENV_12	Envelope #12 4 \276 x 11
ENV_14	Envelope #14 5 x 11 1/2
CSHEET	C size sheet
DSHEET	D size sheet
ESHEET	E size sheet
ENV_DL	Envelope DL 110 x 220mm
ENV_C5	Envelope C5 162 x 229 mm
ENV_C3	Envelope C3 324 x 458 mm
ENV_C4	Envelope C4 229 x 324 mm
ENV_C6	Envelope C6 114 x 162 mm
ENV_C65	Envelope C65 114 x 229 mm
ENV_B4	Envelope B4 250 x 353 mm
ENV_B5	Envelope B5 176 x 250 mm
ENV_B6	Envelope B6 176 x 125 mm
ENV_ITALY	Envelope 110 x 230 mm

Typical Paper Size (Form) Names for Windows 95 and Windows NT Platforms	Description
ENV_MONARCH	Envelope Monarch 3.875 x 7.5 in
ENV_PERSONAL	6 3/4 Envelope 3 5/8 x 6 1/2 in
FANFOLD_US	US Std Fanfold 14 7/8 x 11 in
FANFOLD_STD_GERMAN	German Std Fanfold 8 1/2 x 12 in
FANFOLD_LGL_GERMAN	German Legal Fanfold 8 1/2 x 13 in
ISO_B4	B4 (ISO) 250 x 353 mm
JAPANESE_POSTCARD	Japanese Postcard 100 x 148 mm
9X11	9 x 11 in
10X11	10 x 11 in
15X11	15 x 11 in
ENV_INVITE	Envelope Invite 220 x 220 mm
RESERVED_48	RESERVED--DO NOT USE
RESERVED_49	RESERVED--DO NOT USE
LETTER_EXTRA	Letter Extra 9 \275 x 12 in
LEGAL_EXTRA	Legal Extra 9 \275 x 15 in
TABLOID_EXTRA	Tabloid Extra 11.69 x 18 in
A4_EXTRA	A4 Extra 9.27 x 12.69 in
LETTER_TRANSVERSE	Letter Transverse 8 \275 x 11 in
A4_TRANSVERSE	A4 Transverse 210 x 297 mm
LETTER_EXTRA_TRANSVERSE	Letter Extra Transverse 9\275 x 12 in
A_PLUS	SuperA/SuperA/A4 227 x 356 mm
B_PLUS	SuperB/SuperB/A3 305 x 487 mm
A4_PLUS	A4 Plus 210 x 330 mm
A5_TRANSVERSE	A5 Transverse 148 x 210 mm
B5_TRANSVERSE	B5 (JIS) Transverse 182 x 257 mm
A3_EXTRA	A3 Extra 322 x 445 mm

Typical Paper Size (Form) Names for Windows 95 and Windows NT Platforms	Description
A5_EXTRA	A5 Extra 174 x 235 mm
B5_EXTRA	B5 (ISO) Extra 201 x 276 mm
A2	A2 420 x 594 mm
A3_TRANSVERSE	A3 Transverse 297 x 420 mm
A3_EXTRA_TRANSVERSE	A3 Extra Transverse 322 x 445 mm

Appendix B: Plotter Driver File Syntax

This chapter describes the records that can be included in a plotter driver file.

Introduction

Plotter driver files are ASCII text files that can be edited with a text editor, such as Windows Notepad. Plotter driver files contain records and associated keywords. The maximum record length is 80 characters so each line is a record. Where a record requires more than 80 characters, you can place a backslash (\) at the end of the line and continue on to the next line. The backslash acts as a continuation character. Multiple lines can be included in a single record by placing backslashes at the end of each preceding line. For example:

    weight_strokes(mm)= (0.25, 0.50, 0.75, 1.00, 1.25, 1.50\
    1.75, 2.00, 2.25, 2.50, 2.75, 3.00, 3.25, 3.50, 3.75\
    4.00, 4.25, 4.50, 4.75, 5.00, 5.25, 5.50, 5.75, 6.00)

is read as a single record although it spans several lines.

%include

Lets one plotter driver file include another.

Syntax: `%include "filename"`

where `filename` is the name of the file to be included. Quotation marks enclosing `filename` are optional and may be either quotes (') or quotation marks ("). The filename may include a path or extension. If the path or extension is omitted, the missing values are taken from the `MS_PLTR` configuration variable. Multiple `%include` statements may be used to include multiple files. Included files, in turn, may include other files.

Exceptions

A file cannot include itself nor another file more than once. A file cannot include a file that would cause the inclusion process to loop back on itself. For example, if File A includes File B and File

B attempts to include File A, then File B's `%include` statement is ignored.

Example

Typically, the include file will not be a complete ".plt" file but instead will contain one or more records that can be shared by other "parent" files.

For example, the `style` keyword now supports device-independent units. Therefore, definitions could be moved out of your plotter driver files and into a shared file, which could be included in all your plotter driver files. You then can modify the style definitions for all your plotters by editing this one shared file.

autocenter

Determines whether or not the plot is centered on the page.

Syntax: `autocenter`

Default: if omitted, plots are not automatically centered.

Autocentering occurs only when a new page size or plot rotation is specified.

border

Specifies the type of border to be drawn around the plot boundaries and the information contained in a label placed outside the plot boundary. Also, it determines the default setting of the Plot Border setting in the Plot Options dialog box.

Syntax: `border<options>`

Default: if omitted, no border is drawn.

The following table lists options that are available with the `border` record.

Option	Specifies
`/pen=n`	pen number for border.
`/filename`[a]	that filename be printed on plot.
`/time`[a]	plot current date and time.
`/nooutline`	do not draw border's outline on plot.
`/off`	makes off the default setting of Plot Border in the Plot Options dialog box, while allowing a border to be defined.

Option	Specifies
/comment=("text")[a]	text to be plotted as comment on plot. The comment string is limited to 80 characters.
/text_height=h[a]	how high, in cm, comment text is plotted.
/width[a]	width of border in centimeters.
/font="fontname"	(system printer only) specifies the font to be used.

a. Not supported by all plotters.

Including configuration variables in the comment option

You can include a configuration variable reference in the text of the comment option. For example, if the configuration variable USER was defined as "John Smith" and you defined comment as "User=${USER}," the plotted output would expand the configuration variable so that the text "User=John Smith" appears in the label outside the plot boundary. Note that in this case, braces "{ }" are required around the configuration variable.

Comment strings are limited to 80 characters so care should be used when defining the comment template. There is a chance, also, that a long expansion may get truncated, particularly with the smaller sheet sizes.

Examples

border/pen=1/filename/time

> draws a border around the plot using pen number 1, showing the current time and design file name.

border/pen=1/comment=("My plot")/off

> defines a border and a comment but turns off Plot Border in the Plot Options dialog box by default.

border/pen=1/comment=("User=${USER}")

> defines a border and a comment that includes the configuration variable "USER," which would be expanded to its defined value in the plotted output.

change_pen

Determines if a new pen is selected for each new element color, weight, color and weight or level.

Syntax: change_pen=<criteria>

where: `criteria` can be `color`, `weight`, `both`, or `level`

- If `change_pen=color`, `weight` or `level`, MicroStation automatically sorts through the drawing so that all elements to be plotted with the same pen are plotted one after another. That is, all elements with the same `color`, `weight` or on the same `level`. The pen is then changed and the elements for the next pen are plotted. This assures that the maximum number of pen changes for any plot is equal to the total number of pens minus one.

  If `change_pen=weight,` pens are selected based on line weight. (The line weights to be used with each pen are specified in the `pen=` record.)

- If `change_pen=both`, elements are plotted in the same order in which they were placed in the design. `change_pen=both` should be used for any raster output device such as inkjet, electrostatic and LED. That is, any plotter other than a real pen plotter. Although the design is searched just once (rather than once for each pen), a pen plotter would spend too much time changing pens. The plot looks like the (screen) view plotted, especially when filled shapes overlap. That is, elements placed first lie under overlapping elements placed later, both on screen and on paper.

communication

Defines communication parameters used to generate the plotfile and send it to the plotter.

Syntax: `communication=(keyword1,keyword2,…, keywordn)`

Generally the default parameters are sufficient. However, if the plotter is used with other software that requires different communication parameters, you can use the `communication` record to meet those requirements.

If you find erroneous lines (zingers), incomplete plots or no plotter output at all, make sure the communication parameters in this line of the plotter driver file match the plotter setup.

The following table lists possible keywords, their meanings and their default settings:

Keyword	Meaning	Default

`handshake=n`	Communication method. Allowed values for `n` are `0`, `1`, or `2`, where:
	`0=DIRECT CONTROL`
	`1=XON/XOFF`
	`2=ACK/NAK`[a]
		0

`eom=value`	value for End of Message	3[a]
`numsync=value`	number of sync chars used	1[a]
`syncval=value`	sync code value	2[a]
`checksum`	enable checksumming	no[a]
`recordsize=value`	set record size	1024[a]
`eol1=value`	first end of line terminator	13
`eol2=value`	second end of line terminator	0 (none)
`baud=value`	baud rate	9,600
`port=value`	serial communications port number	1
`parity=keyword`	parity, where `keyword` can be one of: `NONE`, `ODD`, or `EVEN`)	NONE
`databits=n`	number of data bits where `n` can be: 7 or 8	8
`stopbits=n`	number of stop bits, where `n` can be: 1 or 2	1

a. Calcomp 907 format only (least desirable handshake).

Examples

```
communication=(hand=2,eom=13,record=128)
communication=(syncval=22, numsync=1)
```

These lines could be used for a CALCOMP 907 controller set up to use ACK/NAK handshaking, an End of Message character of 13 (carriage return), a record size of 128 bytes, a sync value of 22 (decimal) and one synchronization character.

```
communication=(hand=1, eol1=13, eol2=10)
communication=(port=1, baud=9600, par=none)
communication=(databits=8, stopbits=1)
```

Hewlett-Packard plotters may require lines similar to these, which specify the use of XON/XOFF handshaking, set the first

end-of-line terminator to 13 (carriage return), set the second end of line terminator to 10 (line feed) and communicate through the communications port. The second end-of-line terminator (the line feed) is necessary for certain Hewlett-Packard compatible plotters and may be necessary for transmitting plotfiles to another computer for plotting. MicroStation omits the second end-of-line terminator, by default, to make plotfiles as compact as possible. Line terminators can be disabled entirely by setting both `EOL1` and `EOL2` to 0, allowing the file to become a continuous stream of coordinate data.

default_extension

Specifies the default plotfile extension.

Syntax: `default_extension[/auto_incext]=<fileExtension>`

Default: '000'.

If a numeric default extension is used, such as ".000," and the `auto_incext` option is used, the extension proposed in the Save Plot As dialog box automatically increments with the creation of each additional plotfile.

default_outfile

Specifies the default plotfile name.

Syntax: `default_outfile[/auto_overwrite]=<filename>`

Default: The plotfile is given the same name as the design file but with the extension specified by the `default_extension` record.

If the `auto_overwrite` option is used and a file with the specified name already exists, the file is overwritten by the new plotfile without confirmation.

default_scale

Specifies the default scale factor to generate plots at the same scale every time.

Syntax: `default_scale=<scale factor>`

Default: Largest plot that fits on page. That is, the size and scale factors are set for plots to be the largest plot that fits on the paper size selected.

If the `default_scale` record exists anywhere in the plotter driver file, MicroStation uses this scale factor to set the starting scale when you open the Plot settings box.

driver

Specifies the name of an MDL plotter driver used to stroke the plot. The driver may either reside in the directory pointed to by the MS_MDL configuration variable or in the same directory as the plotter driver file.

Syntax: `driver=<FileName>`

Where `FileName` is the name of the MDL plotter driver to be used. A full path may be specified.

The following table shows the types of printers for which MDL plotter drivers are supplied and for each type, the name of the supplied driver and plotter driver file, as well as the plot resolution. The drivers are in MicroStation's "mdlsys/asneeded" directory. The sample plotter driver files contain the appropriate `driver` records.

end_plot

Specifies action taken by plotter after a plot has been generated.

Syntax: `end_plot=<action>`

Where `action` can be one of the following:

Action	Effect
`eject`	Advances the page after plotting (on plotters that support this feature).
`pause`	Takes the plotter off line to wait for operator intervention.
`beep`	Sounds the bell after stroking is completed.
`replot=n`	For HP-GL/2 devices that support the RP command for plotting multiple copies.

Default: No action.

fence_outline

Specifies whether the outline of the fence should be plotted (if the plot is generated from a fence).

Syntax: `fence_outline</pen=penNumber>`

Where `penNumber` is the pen number to use when plotting the fence outline.

If no `fence_outline` record exists the default is to not draw the fence outline.

font

PostScript devices only.

Specifies the PostScript font name to be mapped to MicroStation font numbers.

Syntax: `font("fontName")=(IGDSfontNumbers)</Kern=n></Sizeratio=m>`

Where:

`fontName` — is the name of the PostScript font to use.

`IGDSfontNumbers` — is the font number or numbers inside MicroStation to map to `fontName`.

`Kern` — is the intercharacter spacing controlled by the parameter `n`.

`Sizeratio` — is a ratio that can be set with the parameter `m` to adjust how large the font is printed (where 1 is the default size).

This is not "what-you-see-is-what-you-get" in the sense that you may see a font in the design file that looks different from the printed output. You can adjust the `Sizeratio` value to make the printed text more like the MicroStation font for which it is substituted. Making the characters taller also makes them wider.

In order for this mapping to work correctly, the font must be resident in the printer prior to sending the plotfile to the printer.

Examples

`font("Helvetica")=(1,2,44-126)/Kern=35`

`font("Courier")=(53,67)/Kern=25`

These lines can be used to map fonts 1, 2, 44-52, 54-66 and 68-126 in MicroStation to the Helvetica font and map fonts 53 and 67 in MicroStation to the Courier font on the PostScript printer.

`font("Helvetica")=(1,2,44-126)/Kern=35/Sizeratio=1.5`

In this case, the Helvetica characters are printed 50% larger than they are in the first example.

hardware_arcs

PostScript devices only.

Specifies to use the hardware arc feature built in to the plotter to plot circles, arcs and ellipses.

Syntax: `hardware_arcs`

hardware_raster and software_raster

Tells MicroStation that the plotter is capable of plotting raster elements (type 87/88 and raster reference).

Syntax: `hardware_raster [options]` or `software_raster [options]` the two records are equivalent.

All raster data is sent first, with vector data plotted on top.

Option	Specifies
`/resolution=`	Defines output resolution in pixels per plotter unit, where the plotter unit is that defined by the `resolution` record.
`/background=`	Defines background color. If `/background=white`, on a monochrome plotter, the image is inverted from what is seen on screen; white on screen is black on paper. If `/background=black`, the image is plotted as seen on the screen. If `/background=color`, a color image can be plotted to HP-GL/2 and color Postscript devices.

largest_polygon

Specifies the largest polygon (in terms of vertices) that a plotter can handle, which varies by model, memory and setup.

Syntax: `largest_polygon=<vertices>`

Practical values for `threshold` (a plotter-dependent value) are 100 for HP-GL plotters and 512 for HP-GL/2 plotters.

linecap

PostScript, HP-GL/2, CGM, Windows EMF/WMF and HPGL/RTL devices only. Refer to the relevant ".plt" files for the valid values.

Specifies the type of line caps to use on plots.

Syntax: `linecap=<value>`

linejoin

PostScript, HP-GL/2, CGM, Windows EMF/WMF and HPGL/RTL devices only. Refer to the relevant ".plt" files for the valid values.

Specifies the type of line joins to use on plots.

Syntax: `linejoin=<value>`

miter_limit

PostScript, HP-GL/2, CGM, Windows EMF/WMF and HPGL/RTL devices only. Refer to the relevant ".plt" files for the valid values.

Specifies the limit for mitered joins on plots.

Syntax: `miter_limit=<value>`

model

Specifies the plotter model.

Syntax: `model=<model_name>`

Where `model_name` can be `7440A, 7470A, 7475A, 7550A, 7580B, 7585B, 7586B, 7595B, 7596B, HPGL2, HPLJET3, DMP52, DMP56, CAL906, CAL907, CAL960, LASERWRITER, POSTSCRIPT,` or `MDL`.

Example

`model=7580B` specifies that a Hewlett-Packard HP7580B is being used.

move_mode

Include this record only if your plotter does not accept relative moves (see your plotter documentation).

Syntax: `move_mode=absolute`

Default: `relative`

If this record is omitted, the value of `move_mode` is assumed to be `relative`.

num_pens

Indicates how many pens the selected plotter can use.

Syntax: `num_pens=<number>`

Where `number` can be a number from `1` to `255` inclusive.

✏ On greyscale devices, setting `num_pens=1` maps all colors to black.

pattern_length

Hewlett-Packard and Calcomp 907 pen plotters only.

Alters the length of one complete pattern that the plotter uses to display a line style. The parameter is represented as a percentage of the diagonal size of the plot.

Syntax: `pattern_length(pattern_num)=<length>`

Where:

`pattern_num` — is the line style number (1-7).

`length` — specifies the length of the pattern as a percentage of the diagonal size of the plot (floating point number between 1-100). Lengths outside the 1-100 range can be used but are not recommended.

Default: Varies according to line style.

If `pattern_length` is omitted, the default values vary according to line style. The most common pattern length is approximately one percent of the diagonal size of the plot.

If the hardware-defined line styles are not satisfactory, your own line styles can be defined in the record for style (page B-20).

pen

Defines the parameters that determine which pens to use to draw individual elements and specify the velocity, force and acceleration of each pen (when possible).

Syntax: `pen(pen_number)=(<colors, weights, or levels>)[options]`

Where:

`pen_number` — is a number between 1 and `num_pens` that specifies which pen is affected by this record.

`colors, weights` or `levels` — is a list of color numbers, weight numbers or level numbers, depending on the keyword specified in the `change_pen` record. (If the keyword specified in the `change_pen` record is `both`, then the `pen` record maps to `color`.)

The `colors` (0-255), `weights` (0-31) or `levels` (1-63) list should be a series of numbers separated by commas, for individual numbers, or dashes to indicate a range of numbers.

If no `pen` records exist in the plotter driver file, the default pen-to-color/weight mapping is established so that the pens are assigned to colors cyclically, with color 0 assigned to pen 1, color 1 assigned to pen 2, and so on. Colors are always assigned to the last pen for which they are specified.

If a `pen` record is too long, it can be split into two `pen` records with the same `pen_number`.

Valid `options` are:

Option	Specifies
`/speed=`	Speed of pen, as a decimal number in cm/s. See plotter documentation for limits on pen speed for your plotter.
`/force=`	Force of pen, as an integer and should be defined in plotter documentation. The HP7470 and HP7475 plotters do not allow force selection and will ignore this parameter.
`/acceleration=`	Acceleration of pen, as an integer and should be defined in plotter documentation. The HP7470 and HP7475 plotters do not allow acceleration selection and will ignore this parameter.
`/rgb=`	Specification of pen color using the RGB color model. If you are using a pen plotter, the RGB option is desirable for on screen plot preview. If you are using a plotter with programmable color, the RGB option lets you override the screen color when plotting.

If pen speed, force or acceleration is omitted, the values are assumed to be set up in the plotter (normal case).

Examples

`pen(4)=(1,3,20-33,44)`

> Specifies that colors (assuming that color is the pen change criteria) 1, 3, 20 through 33 and color 44 are to be drawn with pen number 4.

`pen(1)=(1-128)`

`pen(2)=(4,7,20)`

> Specifies that elements with colors 4, 7 and 20 are drawn using pen 2 and all others are drawn using pen 1.

```
pen(2)=(6)/rgb=(100,100,100) ;medium grey
```

Specifies that color 6 (assuming that color is the pen change criteria) is to be mapped to `rgb` values of `100,100,100` and drawn using pen 2. It is good practice to add a comment such as shown here (`;medium grey`) describing the assigned color.

pen_change

Specifies an action to be taken by the plotter after every pen change operation.

Syntax: `pen_change=<option>`

Where `option` can be:

`eject` — advances the page between pen changes (on plotters that support this feature).

`pause` — takes the plotter off line to wait for operator intervention.

Default: No action taken.

pen_width

Determines the distance between multiple strokes on a weighted line, expressed in units of resolution. Also used to space the lines when filling a polygon or filled text.

For example, if the plotter resolution is 0.025 mm (the default) and `pen_width` is set to 10, then the distance between strokes for a multi-stroke line is 0.25 mm.

Syntax: `pen_width(pen_number)=<width>`

Where:

`pen_number` — is a number between 1-`num_pens` indicating the pen for which the width is to be specified.

`width` — is an integer (0 -15) specifying the width of the line in plotter resolution steps, if the plotter can draw wide lines. If the plotter cannot draw wide lines, `width` specifies the number of plotter resolution steps separating the strokes specified in the `weight_strokes` record.

If omitted, the default value is 9

pen_separation

Calcomp 960/965 only.

Tells the plotting subsystem the distance (0.0-36.0 inches) between pens on Calcomp 965 plotters (to compensate for the pen change on this plotter).

Default: 0.60 inches

When the plotter selects another pen, it moves the pen carrier but never updates its own internal pen position. This record should be used only in cases where offset errors are occurring on Calcomp 960 format plotters with more than one pen.

pentable

Specifies a pen table file to be loaded when the plot driver file is selected.

Syntax: `pentable=\dir\file.tbl`

where `\dir\file.tbl` are the directory and the name of the pen table file to be loaded.

point_size

Enables plotting of points and determines size of plotted points.

Syntax: `point_size=<cm>`

Where `cm` is a size in centimeters (or decimal fractions of a centimeter).

Representation of points is determined by the device driver. The HP-GL and HP-GL/2 drivers represent points as filled circles; the Calcomp 906/907 drivers represent them as circles.

prefix

Specifies the name of a file, the contents of which will be added to the beginning of a plotfile.

Syntax: `prefix=<file_spec>`

Where `file_spec` is the name of the file to be added to the plotfile. Include a full path if the file is not in the same directory as the plotter driver file that specifies them.

Default: if omitted no prefix file is used.

Example

```
prefix=setswitch.dat
```

> Defines a file named `setswitch.dat` that could contain escape sequences defining the plotting language used by a plotter.

prolog

PostScript only.

Specifies the PostScript prolog file, the content of which is copied by MicroStation into PostScript plotfiles.

Syntax: `prolog=<file_spec>`

Where `file_spec` defines the file to be copied into the PostScript plotfile. `file_spec` can include a full or partial path specification. In the absence of a full path specification, MicroStation uses the `MS_PLTR` configuration variable definition to complete the specification.

Default: "pscript.pro" in MicroStation's "plotdrv" directory.

resolution

Specifies the resolution of the plotter and the units in which distances on the plotter are expressed.

Syntax: `resolution(units)=(<x_resolution>,<y_resolution>)`

Where:

`units` — can be `mm`, `cm`, `dm`, `m`, `in`, or `ft`.

`x_resolution` and `y_resolution` — are floating point numbers. They specify the height and width of a single pixel on the device, expressed in the specified units.

For example, for a 300 dpi device:

```
resolution(IN)=(0.003333333333333333,0.00333333333333333
3)
```

because each pixel is 1/300=0.003 inches high/wide.

The `resolution` parameter is defined by the plotter manufacturer and is used to calculate the number of plotter steps per plotter unit.

On Hewlett-Packard plotters the resolution is 0.025 mm/PLOTTER UNIT. To express this value in units other than mm, simply divide the new unit by its equivalent in mm and multiply the result by 0.025.

For example, to use inches as the units for your plotter, where 25.4 mm is equivalent to 1 inch, use

```
1/25.4 (IN/MM) * 0.025 = (0.000984252 INCHES/PLOTTER
UNIT).
```

✍ There is a connection between the `resolution` and `hardware-raster` records. `hardware_raster/resolution=nnn` defines the resolution in terms of pixels per unit, where unit is defined in the `resolution` record. For example, if your ".plt" file contains:

```
resolution(IN)=(…)
```

```
hardware_raster/resolution=80
```

then 80 dots per inch raster is generated.

If you then switch to metric units without updating the `hardware_raster/resolution` record, such that:

```
resolution(MM)=(…)
```

```
hardware_raster/resolution=80
```

then 80 dots per millimeter raster is generated, which is a much higher resolution and will result in larger plotfiles.

roll

PostScript devices only.

Orientation setting for roll-type media.

Syntax: `roll=<value>`

Where `Value` can be:

Value	Meaning
0	Normal default orientation for the specified page size
1	Rotate image 90° counterclockwise
2	Rotate image 180° counterclockwise
3	Rotate image 270° counterclockwise

rotate

Indicates the direction in which plots are automatically rotated to maximize the area on the page used for the plot.

Syntax: `rotate=<direction>`

Where `direction` can be:

`none` — no rotation.

cw — clockwise rotation 90°.

ccw — counterclockwise rotation 90°.

size

Specifies the plotting area of the paper size supported by the plotter.

Syntax: `size=(<width>,<height>)/offset=(<woff>,<hoff>)/number=n`
`/scale=n/name=<name>/manual_origin=<origin option>`

Where:

`width` and `height` — are the width and height of that part of the paper used for the plot. `width` specifies the part of the page used along the X (usually the longest axis of the paper) and `height` is the portion of the page to be used along the Y (usually the shortest axis). Equally, it is possible to have a `width` value that is smaller than the `height` value.

Other options specify the following:

Option	Specifies
/offset=	Sets the distances from the paper origin to the lower left corner of the plot in plotter units (defined in the `resolution` record). `woff` and `hoff` — sets the distance along the X (width) and Y (height) axes, respectively.
/number=	Paper size number (on HP7475A and some HP-GL/2 devices) to specify paper size. (The HP7470 uses only one paper size and the other plotter models sense the paper size automatically.) Use /number=0 for either B or A3 paper or /number=4 for A or A4 paper. or Form length, in inches, for some MDL-driven, ESC/P-language plotters, such as the Canon BJ230, that accommodate ledger size paper but lack a DIP switch for turning off the "page break skip" feature.
/name=	Name to specify the paper size. Names can be up to eight characters long and each name should be unique. Each `size` record must have a /name= keyword.

Option	Specifies
`/manual_origin=` (Hewlett-Packard plotters only)	Manually sets the plot origin: • `p1` — Sets lower left corner • `p2` — Sets upper right corner • `both` — Sets both corners If either `p1` or `p2` is used, the size of the drawing is determined by the size and scale factors in the Plot Layout dialog box. The position on the page is determined by the setting of the `p1` or `p2` point on the plotter. If the `both` option is used, MicroStation fits the plot between the `p1` and `p2` points set up on the plotter at plot time. The size and scale factors in the Plot Layout dialog box are irrelevant.
`/scale=`	Sets the default plotting scale factor for a particular paper size.

The paper origin is the (0,0) point for the plotter. The position of the origin varies with the plotter type. For example, Hewlett-Packard 75XX plotters measure the paper when the "Chart Hold" button is pressed and they assign the paper origin to the exact center of the page.

Between 1-5 `size` records must be in the plotter driver file. The last `size` record in the file becomes the default paper size.

For example, suppose you are using an HP7585B plotter and a 34 × 44 inch piece of paper. Since the hard clip limits of the plotter are approximately 0.5 inches on each side, the maximum usable area of the paper is approximately 33 × 43 inches. Consider defining three paper sizes (one at the full size (33 × 43) and two at half size (33 × 21.5) on the left and right halves of the paper). To do this, place the following lines in the plotter driver file:

```
size=(21.5,33)/offset=(0,-16.5)/num=1/name=right
size=(21.5,33)/offset=(-21.5,-16.5)/num=1/name=left
size=(43,33)/offset=(-21.5,-16.5)/num=1/name=full
```

If you were using a Calcomp 1043, the same paper sizes would appear in the plotter driver file as:

```
size=(21.5,33)/offset=(22,.5)/num=1/name=right
size=(21.5,33)/offset=(.5,.5)/num=1/name=left
size=(43,33)/offset=(.5,.5)/num=1/name=full
```
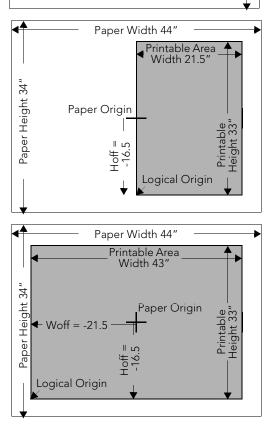
The following diagram illustrates the results of the above settings.

Sample plotting areas.

Top to bottom: Plot on left side, right side, and on full sheet (diagrams not to scale).

software_raster

See hardware_raster and software_raster (page B-9).

start_plot

Specifies an action to be taken by the plotter before the plot is generated.

Syntax: `start_plot=<option>`

Where option can be:

`eject` — advances the page before plotting (on plotters that support this feature).

`pause` — takes the plotter off-line to wait for operator intervention.

Default: No action taken

stroke_tolerance

Computes the number of vectors used when stroking arcs and circles, where 10 represents the greatest number of vectors.

Syntax: `stroke_tolerance=<value>`

Where `value` is a figure between 1 and 10 (inclusive)

Default: 3.5

Consider the trade-off between plotting time and plot quality when choosing a `stroke_tolerance` value. In most cases, the default value of 3.5 is sufficient.

style

Used to define line styles.

Syntax: `style(style_number) = (<pen_down,pen_up>)/option/ units=MM,IN,or DOTS (the default)`

Where

`style_number` — is the line style number (in the range 0 and 7), which corresponds to the MicroStation line code number.

`pen_down`, `pen_up` — these pairs are distances represented in specified units or in plotter resolution units.

`/units=MM` or `/units=IN` specifies that the `pen_down`, `pen_up` pairs are in millimeters or inches respectively. Where `/units=DOTS` or when `/units` is not specified, then distances are in plotter units, which vary for different plotters.

Default: Varies according to plotter.

If a plotter can generate user-defined line styles, the plotting subsystem downloads a pattern to the plotter. Several plotters support this feature. For other plotters, use the `/nohardware` option in the `style` record.

Line styles are a series of pen down, pen up distances that define one repetition of the pattern being drawn. Line styles can be up to 11 inches long and can have up to 5 pen down, pen up pairs.

Valid values for `option` are:

`/hardware` — indicates that a plotter is capable of generating custom line styles. When this option is used, the plotting subsystem downloads the line style pattern to the plotter. This is the default value if no option is specified.

`/nohardware` — indicates that a plotter does not support the generation of custom line styles.

Example

Using the `/units` setting makes the task of defining line styles much easier than using plotter resolution units. Take, for example, a Calcomp 907 plotter with a resolution of 2032 lines per inch. If `/units` is set to DOTS or left out completely, then definitions are in plotter resolution units. To define style number 2 as a 1/2 inch line, followed by a 1/4 inch space, followed by an 1/8 inch line, and then followed by a 1/4 inch space, in plotter resolution units, the `style` record would read:

```
style(2)=(1016, 508, 254, 508) /nohardware
```

On HP plotters with a resolution of 1016 lines per inch using plotter resolution units, the record would read:

```
style(2)=(508, 254, 127, 254)/nohardware
```

If the `/units` option is set to `IN` for both plotters, then the record is the same for both plotters and is much simpler to understand:

```
style(2)=(0.5, 0.25, 0.125, 0.25) /nohardware/
units=IN
```

suffix

Specifies the suffix file, the content of which is added to the end of the plotfile created by MicroStation. In the absence of a full path specification, MicroStation looks for the suffix file in the same directory as the plotter driver file.

Syntax: `suffix=<file_name>`

Where `file_name` is a file that is added to the end of the plotfile.

Default: if omitted no suffix file is used.

To accommodate networks, plotter features and other aspects of a particular site or project, you can add data files to the beginning or end (or both) of the plot data created by MicroStation drivers. Two keywords permit this:

```
prefix=<filename_1>
```

```
suffix=<filename_2>
```

If these records are present, the contents of the named files (`filename_1` and `filename_2`) will be added to the plotfile. Include a full path if the files are not in the same directory as the plotter driver file that specifies them.

Some PostScript printers require that the data stream end with a <Ctrl-D>. The sample file "controld" (installed in MicroStation's "plotdrv" directory) contains the single character <Ctrl-D>. To add this character to the plotfile, you can add to the PostScript plotter driver file the line `suffix=controld`.

sysprinter

(For Windows platforms only) Designates a specific system printer, form, orientation and offset. These options can be used alone or in combination.

Syntax:

```
sysprinter  /name="printer-name"
    /form=f
    /orientation=o
    /offset=(h,v)
```

Where:

```
/name
```

Specifies the printer name. To obtain a valid printer name, first open the StartMenu > Settings > Printers folder. Below the icons for each printer is the printer name. The name will take one of two forms:

1. A local printer name like `HP LaserJet 4 Plus`.

In this case, the printer name used is just the local name exactly as seen in the Printers folder:

```
/name="HP LaserJet 4 Plus".
```

2. A network printer name like `hp755cm on printserver`, which specifies a printer share name and a network computer.

In this case, the printer name used is specified as `\\computer\share`. For example:

`/name="\\printserver\hp755cm"`

If necessary, you can find the computer name needed by looking at StartMenu > Settings > ControlPanel > Network.

If the printer name contains spaces or '/', it should be enclosed in either single (') or double (") quotes'.

The /name qualifier is optional. If omitted, the Windows default printer will be used.

/form

This value can be any valid form name for the that printer, such as letter, legal and so on. For a list of all standard form names plus how to check for valid form names for your printer, refer to "Paper size (Form) names for Windows platforms" on page A-8.

Note: Some manufacturers have used form names which contain characters outside the normal ASCII printable range. Example: for a form name like 8 1/2 x 11, instead of specifying 3 characters 1, /, and 2, they have put in the code for a special one-half symbol which only takes up one character position.

To handle this case, you can enter "ANSI 8?x11" and MicroStation will match the actual name. If the form name contains spaces or '/', it should be enclosed in single (') or double (") quotes. If the form name contains " to indicate inches, use a single quote pair, such as:

`/form='letter 8 1/2" x 11"'`

The `/form` qualifier is optional. If omitted, the Windows default setting for the named printer will be used.

/orientation

This value can be either `portrait` or `landscape`.

The `/orientation` qualifier is optional. If omitted, the Windows default setting for the named printer will be used.

/offset

Leaves horizontal and vertical space for the border text. For example, if your units are set to inches, a setting of (0.00, 0.25) will leave a 1/4 inch space at the bottom of the plot so border text may be drawn.

The `/offset` qualifier is optional. If omitted, no space will be left.

weight_strokes

Specifies the mapping of MicroStation line weights to line thicknesses in the plot.

Syntax: `weight_strokes(units)=(list)`

where:

`units` — can be `mm`, `in` or `dots` (the default) to specify that line thicknesses are plotted in millimeters, inches or plotter resolution units, respectively.

`list` — is a list of line thicknesses, expressed in millimeters, inches or dots (plotter resolution units). For example the following entry:

**weight_strokes(mm)=(0.25, 0.50, 0.75, 1.00, 1.25, 1.50, 1.75, 2.00, **

    **2.25, 2.50, 2.75, 3.00, 3.25, 3.50, 3.75, 4.00, **

    **4.25, 4.50, 4.75, 5.00, 5.25, 5.50, 5.75, 6.00, **

    6.25, 6.50, 6.75, 7.00, 7.25, 7.50, 7.75, 8.00)

Lists line thicknesses in millimeters (mm) for each MicroStation line weight from 0 through to line weight 31. Each line thickness is separated by a comma. Because the entry spans more than one line, backslashes are placed at the end of preceding lines to tell the system to continue reading the record on the following line.

Where no units are specified or when `dots` is specified as the units, then the line thicknesses are based on plotter resolution units. The listed values specify the number of strokes that are used for each line weight when line weight is enabled. Thus, printers with different resolution units will require different settings to produce the same line thickness. If omitted, the default value is the line weight divided by 2.

Example

If you used the following record:

```
weight_strokes(mm)=(0.25,0.50,0.75,1.00,1.25,1.50,1.75,2.00)
```

then line weight 0 is plotted at a thickness of 0.25mm, line weight 1 is plotted at a thickness of 0.50mm, and so on. This record is valid for any plotter because the units are specified as millimeters.

When `dots` is specified as the units or if units are omitted, the stroke thickness for a line weight in millimeters is computed by multiplying the corresponding `weight_strokes` value by the plotter's resolution units. If, for example, the plotter resolution units are 0.025, then the following:

```
weight_strokes=(10,20,30,40,50,60,70,80)
```

causes line weight 0 to be stroked ten times to produce a line thickness of 0.25 mm, line weight 1 to be stroked twenty times to produce a line weight of 0.50 mm, and so on. To produce these exact line thicknesses, this record is valid only for plotters having a resolution of 0.025mm.